
Table of Contents

Chapter 1 : Introduction	1
What's included with BOARDMAKER	1
How to use this manual	1
How to contact Tsien	2
Chapter 2 : Fundamentals	3
BOARDMAKER and your computer system	3
Program Capabilities	4
The READ.ME file	6
Installation	6
Hard disk	6
Floppy disk	7
Starting BOARDMAKER	7
BOARDMAKER conventions	7
Function keys	8
Shift, Alt and Control keys	8
Using Menus	8
Selection banner	9
The BOARDMAKER main menu	9
The BOARDMAKER screen	10
Messages and line editor	10
CGA and HGA display limitations	12
Menus or keystrokes ?	12
The Option menu	12
Chapter 3 : User Guide	13
A Quick Tour	13
The Anatomy of Tracks	15
Nets and netlists	15
Design rules	17
The Current Mode	17

The Current Item	17
Selecting tracks	18
The Option Menu and Information Banner	18
Track Mode	18
Pad Mode	19
Text Mode	19
Symbol Mode	19
Block Mode	19
Net mode	19
The Status Banner	19
Layout, schematic or library editor ?	20
How to get the most out of BOARDMAKER	21
Net list creation	21
Communicating with a schematic capture package	21
Creating you own net list	22
Placing the components	22
Routing the board	23
Changing your design	23
Which layers to use	24
Checking the entire board	24
Using the Full design rule checker (Full DRC)	24
Exporting a netlist	25
Reports	25
Board manufacture	25
Matrix printer plots	25
Laser	26
Penplot	26
Photoplot	26
Numerically controlled (NC) drilling data	26
Placing Items and Associated Activities	27
Placing tracks	27
Track width	28
All width	28
Segment width	28
Changing layers	29
Segment layer	29
Vias, tracks and layer pairs	29

All layer	30
Angle assist	30
Assist flip	31
Placing pads	31
Pad layer	31
Pad shape	31
Pad width	32
Fixing the pad	32
Placing text	32
Text size	33
Text width	33
Text layer	33
Incrementing and decrementing text	34
Fixing the text	34
Placing components or symbols	34
Symbols vs Macros	35
Placing a symbol	35
Fixing the symbol	36
Placing circles and arcs	36
Creating an arc	37
Editing Items	37
Editing a track or track segment	37
Selecting the track	38
Changing layers	38
Changing track widths	38
Moving a complete track	38
Splitting a track	38
Moving track segments	39
Inserting track segments	39
Editing pads	40
Changing layer, width or shape	40
Moving the pad	40
Editing text	40
Selecting the text	41
Changing layer, width and size	41
Rotating text	41
Mirroring text	41
Changing the textual content	42
Moving the text	42

Editing a symbol	42
Moving a symbol	42
Changing the designator	43
Moving the designator	43
Rotating the designator	43
Hiding and showing the designator	44
Changing the designator size and width	44
Giving a component a value	44
Rotating a symbol	44
Mirroring a symbol	44
Deleting items	45
Deleting tracks	45
Delete a track segment	45
Delete a whole track	46
Deleting pads	46
Deleting text	46
Deleting a symbol	47
Block manipulation	47
Defining the block	47
Drawing the block	47
Changing the block origin	48
Defining which items are affected	48
Maintaining connectivity	49
Deleting a block	49
Moving a block	49
Repeating a block	50
Rotating a block	50
Clockwise	50
Anticlockwise	51
Mirroring a block	51
Printing or plotting a block	51
Creating a ground plane	52
Repeating items	53
Repeating tracks	53
Repeating pads	53
Repeating text	54
Repeating symbols	54

Current settings	54
Changing the track and pad sizes	54
Layer and colour selection	55
Fixed and floating origins	56
Cursor settings	56
Auto vias	56
Layer pairs	57
Inches or millimetres ?	57
Grid sizes and settings	57
Visible grid	57
Snap grid	58
Nearest pad and node snap	58
Net highlighting	58
Highlighting a connection	58
Highlighter warnings	59
Deleting a highlighted net	60
Using and managing netlists	60
Mode 1 - Using a schematic capture program	60
Mode 2 - Using the built-in net list editor	61
Mode 3 - On the fly design with global checking	61
Mode 4 - On the fly design with manual checking	62
Loading a netlist	62
Using the netlist editor	63
Adding a net	64
Editing a net	65
Moving along the net	65
Deleting nets and net nodes	66
Changing the net name	66
Changing the design rules for a net	66
Viewing nets and nodes	67
Clearing a netlist	67
Create nets	67
Exporting the nets	68
Hiding and showing nets	69
Optimizing nets	69
Design rule checking	69
Setting the global design rules	70

Applying the design rules	70
Viewing the violations	71
Using the Full design rule checker	72
Moving around your drawing	72
Zoom and Unzoom	73
Pan	73
Tagging	73
Drawing information	73
Status window	74
Usage window	74
Managing your PCB and schematic files	74
Loading and saving files	75
Directory	75
Clearing the drawing area	75
Defining the working directories	75
The Backup timer	76
Using and organizing symbol libraries	76
Symbol library editor	76
Browsing libraries	77
Viewing a library	77
Creating a new symbol	81
Creating a new library	81
Erasing a symbol	82
Changing an existing symbol	82
Changing the configuration	83
X and Y mouse locking	83
Mouse settings	83
Message timer	84
Config auto save	84
Keyboard settings	84
Output facilities	84
Dot-matrix printer	85
Layers to print	85
Printer settings	86
Output mode	86
Resist clearance	86

Group layers	86
Output style	86
Laser printers	86
Printer settings	86
Penplotter	86
Drawing details	87
Current plot setup	87
Plotter settings	87
Plot setup	88
Choosing the layer to plot	88
Output to	88
Save setup	89
Photoplotter	89
Drawing details	89
Plot status	90
Solder resist mask	90
Layer to plot	90
NC drill	90
Drawing details	90
Drill sizes	91
Changing the hole sizes	91
Restore hole sizes.....	91
Chapter 4 : Command Reference	93
THE COMMAND REFERENCE	95
ADD DESIGNATOR	97
ADD NET	99
ADD PAD	102
ADD REFERENCE	103
ADD SYMBOL	105
ADD TEXT	106
ADD TRACK	107
ADD TO LIBRARY	108
ALIAS	110
ALL LAYERS	111
ALL WIDTH	113
ANGLE ASSIST	114

ANTICLOCKWISE Designator 115

ANTICLOCKWISE Designator prefix 116

ANTICLOCKWISE Text 117

ANTICLOCKWISE Symbol 118

ANTICLOCKWISE Pad 119

ANTICLOCKWISE Block 120

ARC / CIRCLE 122

ASSIST FLIP 124

AUTO PAN 125

AUTO VIA 126

BACK NET 127

BACK Net node 128

BACK Track node 129

BACKUP TIMER 130

BLOCK 131

BLOCK OPTIONS 132

BLOCK ORIGIN 133

BROWSE LIBRARIES 135

CLEAR NETS 136

CLEAR VIOLATIONS 137

CLOCKWISE Block 138

CONNECT MODE..... 140

CREATE NETS 141

CURRENT Designator prefix 142

CURRENT Net 143

CURRENT Pad 144

CURRENT Reference 145

CURRENT Symbol 146

CURRENT Text 147

CURRENT Track 148

CURSOR BOX 149

CURSOR MODE 150

CURSOR SIZE 151

CURSOR TRAVEL 152

CURSOR TYPE 153

DECREMENT Alias 154

DECREMENT Name 155

DECREMENT Text 156

DELETE Net node 157

DELETE Track 158

DESIGNATOR POSITION 159

DESIGN RULES Global 160

DESIGN RULES Net based 161

DESIGN RULE CHECK (DRC) 162

DIRECTORY 164

DIRECTORY SETUP 165

EDIT DESIGNATOR 167

EDIT NET 168

EDIT PAD 170

EDIT REFERENCE 171

EDIT SYMBOL 172

EDIT TEXT 173

EDIT TRACK 174

END Net node 175

END Track node 176

ERASE SYMBOL 177

EXIT 178

EXPORT NET 179

EXPORT SYMBOL 180

FLOOD FILL 181

FORCE LAYER 185

FULL DESIGN RULE CHECK..... 186

GOTO ORIGIN 188

GRID 189

Table of Contents

GRID SNAP	190
HIDE/SHOW DESIGNATOR	192
HIDE NETS	193
HOME Net node	194
HOME Track node	195
INCREMENT Alias	196
INCREMENT Name	197
INCREMENT Text	198
INSERT/MOVE NODE	199
INVERT ORDER	201
KEYBOARD COUNTRY	203
KEYBOARD LAYOUT	204
KILL Block	205
KILL Designator prefix	206
KILL HIGHLIGHTED NET	207
KILL Net	208
KILL Pad	209
KILL Reference	210
KILL Symbol	211
KILL Text	212
KILL Track	213
LAYER Designator prefix	214
LAYER Pad	215
LAYER PAIRS	216
LAYER Text	217
LAYER SELECTION	218
LOAD	219
LOAD NET	220
MESSAGE TIMER	223
MIRROR Block	224
MIRROR Designator prefix	226
MIRROR Pad	227

Table of Contents

MIRROR Symbol	228
MIRROR Text	229
MOUSE LOCKING	230
MOUSE SETTINGS	231
NAME NET	232
NAME Reference	233
NC DRILL	234
NET HIGHLIGHT	235
NEAREST NODE	237
NEAREST PAD AND NODE SNAP	238
NEW	239
NEXT NET	240
NEXT Net node	241
NEXT Track node	242
NEXT VIOLATION	243
OPTIMIZE NETS	245
PAN	246
PENPLOT	247
PENPLOT BLOCK	250
PHOTOPLOT	251
PREVIOUS VIOLATION	254
PRINT ALL	255
PRINT BLOCK	257
PRINTER SETTINGS	258
QUASI-TRACK	261
REPEAT Block	263
REFERENCE BOX	264
REPEAT Reference	265
REPEAT Track	266
REPEAT Pad	267
REPEAT Symbol	268
REPEAT Text	269

Introduction

Welcome to the BOARDMAKER user's guide. This manual is a comprehensive guide to BOARDMAKER, an ultra-fast, full-featured and easy-to-use PCB layout and schematic design program written by Tsien (UK) Limited, a British company which specializes in developing CAD products for printed circuit boards.

What's included with BOARDMAKER

BOARDMAKER is designed for computer users with all levels of experience, from beginners who have been using tape-up for circuit board design just moving onto CAD design tools, to experienced users who want complete control over the way a circuit board is laid out - you tell the program where you want the tracks and pads by making selections from intuitive menus or by using direct keyboard commands. You are able to design PCB layouts or schematic diagrams using powerful graphical editors and take them through to the output of the final artwork ready to pass on to your PCB manufacturer - all using one integrated software package with the minimum of fuss and learning.

This manual was written for those without extensive experience of printed circuit board CAD software. It includes a guide to interactive use and a full command reference section which allows you to determine the amount of reading required before you start designing your first layout. The software and manual are designed so you don't feel overwhelmed by a vast quantity of information, but at the same time still provide details of all the facilities available to you.

How to use this manual

The BOARDMAKER user's guide is a complete guide to interactive use on your computer. You can use it both for learning and for reference. You are

Table of Contents

REPORTS	270
RUBBERBAND	271
SAVE	272
SAVE OPTIONS	274
SEGMENT LAYER	276
SEGMENT WIDTH	277
SET ORIGIN	279
SHAPE	280
SHIFT Block	281
SHIFT Track	283
SHOW NETS	284
SIZE Designator	285
SIZE Designator prefix	286
SIZE Reference	287
SIZE Text	288
SPLIT	289
TAGS	290
TEXT CONTENT	291
TEXT CONTENT Designator prefix	292
TEXT CONTENT Designator	293
TRACK AND PAD SIZES	294
UNITS	296
UNZOOM	297
UPDATE SYMBOLS	298
VIEW NET NODE	299
VIEW LIBRARY	301
WIDTH Text	302
WIDTH Designator	303
WIDTH Designator prefix	304
WIDTH Pad	305
ZOOM	306

advised to read the user's guide to understand the options and operations available and then use the command reference when you need more details of a particular command or facility.

If you are an experienced computer or CAD user and you decide to delve into BOARDMAKER's features straight away, make sure you read Chapter 2 (Fundamentals), which contains information about installation and basic use. Subsequent chapters assume you have read Chapter 2.

How to contact Tsien

If you experience any difficulties or you simply have questions or comments please contact the Tsien technical support department.

TSIEN (UK) LIMITED

Cambridge Research Laboratories

Huntingdon Road

CAMBRIDGE

CB3 0DJ

UNITED KINGDOM

Telephone : 0223-277777

Please have the following information handy before you call :

- ☐ User registration number
- ☐ Product version and release number
- ☐ Computer make and model
- ☐ Details of the output devices you are using
- ☐ Operating system version

Remember, to automatically receive product upgrades and receive important announcements, please send your user registration form to the above address as soon as possible - it will also help us to deal with any difficulties more quickly.

Chapter 2

Fundamentals

BOARDMAKER and your computer system

In order to use BOARDMAKER on your computer you need :

- ☐ an IBM PC, XT, AT or 100% compatible computer
- ☐ either one hard disk and at least one floppy disk drive OR no hard disk and two floppy disk drives
- ☐ at least 512K bytes of system memory (RAM)
- ☐ DOS 2.0 or later
- ☐ a compatible Hercules graphics adaptor (HGA), colour graphics adaptor (CGA), multi-colour graphics adaptor (MCGA), enhanced graphics adaptor (EGA) or video graphics array (VGA) adaptor card and monitor. BOARDMAKER auto-detects your adaptor and selects the relevant screen driver.
- ☐ at least one parallel printer port
- ☐ optionally, an Epson FX/MX/LX 9 pin dot matrix printer or an Epson LQ 24 pin printer or 100% compatible
- ☐ optionally, a bus or serial mouse
- ☐ optionally, an HP Laserjet II 300dpi laser printer or 100% compatible
- ☐ optionally, a Graphtec, Roland or Houston pen plotter or other supporting HP-GL or DM-PL languages
- ☐ optionally, access to a Gerber format photoplotter
- ☐ optionally, access to an Excellon format NC drilling machine

These last two items are more likely available via a photoplot bureau and PCB manufacturer respectively.

Program Capabilities

- ☐ Integrated PCB and schematic editor
- ☐ Integrated graphical net list editor
- ☐ Layers - 8 circuit layers, 2 silk screen layers
- ☐ Max board or schematic size - 17 x 17 inches
- ☐ Display support - HGC, CGA, MCGA, EGA and VGA
- ☐ Layer colours - user definable
- ☐ Track widths - 8 from 128 widths in the range 2 to 531 thou.
- ☐ Pad widths - 16 from 128 widths in the range 2 to 531 thou.
- ☐ Tracks - 8 different sizes per layout - 5000 tracks per board with up to 12000 track segments
- ☐ Pads - 16 different sizes per layout - 2000 pads per board excluding those within symbols
- ☐ Pad shapes - 14 different pad shapes including circular, square, oval and rectangular
- ☐ Text - 8 sizes in the range 25 to 400 thou high - 4 text widths in the range 2 to 27 thou- up to 10000 characters per layout or schematic. Text can be moved, rotated, repeated or mirrored
- ☐ Symbols - 1000 symbols per layout of 100 different types. Symbols can be moved, rotated, repeated and mirrored
- ☐ User definable symbol and macro library facilities including a symbol library editor
- ☐ Graphical library browse facility
- ☐ Symbol library independent PCB design files - local copies of the symbols are held in the design files which can be updated and exported as required

- ☐ Nets - up to 800 nets per PCB file
- ☐ Design rule checking (DRC)- checks the clearances between items on the board
- ☐ Real-time DRC display - when placing tracks you can see a continuous graphical display of the design rules set
- ☐ Routing validation - checks a selected route matches with its net
- ☐ Full, net based, design rule checking (NB-DRC) - performs a board-wide verification of the whole layout with extensive error reporting
- ☐ Bi-directional net format translators and symbol/component mapping - allows easy communication with schematic capture packages
- ☐ Automatic component placement via net list import
- ☐ Grid - Separate visible and snap grid - 7 placement grids in the range 2 thou to 0.1 inch
- ☐ Auto via - vias are automatically placed when you switch layers - layer pairs can be assigned by the user
- ☐ Blocks - groups of tracks, pads, symbols and text can be block manipulated using repeat, move, rotate and mirroring commands. Connectivity can be maintained if required
- ☐ SMD - full surface mount components and facilities are catered for, including the use of the same SMD library symbols on both sides of the board if required
- ☐ 45 and 90 degree corners can be automatically introduced
- ☐ Circles - Arcs and circles up to the maximum board size can be drawn. These can be used to generate rounded track corners - this is important for some RF applications
- ☐ Ground plane support - areas of copper can be filled to provide a ground plane or large copper area. This will automatically flow around any existing tracks and pads
- ☐ User selectman backup timer to remind you to save your work
- ☐ User definable message timer to determine the time system messages are displayed. This can be turned off for expert users

- ☐ Output can be sent to a 9 or 24 pin dot matrix printer or an HPGL or DMPL penplotter.
- ☐ Another output option is to generate Gerber format data, which can be sent to a photoplot bureau and an NC drill (Excellon format) file.

The READ.ME file

It is most important that you take the time to look at the READ.ME file on System Disk 1. It contains any last minute information and errata not contained in the manual along with a list of the files which are on each of the System disks.

To read this file, you can either load it into a word processor or print it directly by typing 'PRINT A:READ.ME'

Installation

Before you run BOARDMAKER, you are advised to make copies of the disks provided in the package and store the originals in a safe place. There is no copy protection and you are free to make as many copies of these disks as you usefully need, but the main program will only run if the protection device or 'dongle' is inserted in your parallel port. This device is described in more detail later in this chapter. Use the DOS 'Diskcopy' command to copy the disks as required. If you are not familiar with this DOS command, refer to your DOS reference manual before starting to install or run BOARDMAKER.

Hard disk

The BOARDMAKER installation procedure is designed to ease your path through the installation process. All you need to do is to follow the instructions which appear on the screen at each stage - make sure you read each prompt CAREFULLY.

- ☐ Turn on your computer or exit the program you are running so that you can see the DOS system prompt C:\ on the screen.
- ☐ Insert System Disk 1 in drive A:
- ☐ Type A: and press Enter

- ☐ Type INSTALL and press Enter

Now follow the on screen instructions which are displayed. When the process is finished, you are ready to start using BOARDMAKER.

Floppy disk

Put System Disk 1 in drive A: and System Disk 2 in drive B:. Arrange that the DOS PATH has been set to look at both drive A: and drive B:. As you use the program you will be prompted at various times to insert other relevant disks in the drives.

Make sure you are using copies of the disks provided in the package, not the originals.

Starting BOARDMAKER

Put the BOARDMAKER dongle in any of the parallel ports on your computer. This device must be present for the program to run. It doesn't matter which port you put it in because BOARDMAKER will automatically detect it. Simply put it in the port you want and plug any device which uses the port into the other end. This should not affect your use of the port, it is transparent as far as other software is concerned and can be left in place even if BOARDMAKER is not being used.

You are now ready to start BOARDMAKER. Make sure you are logged onto the drive and directory where you installed the BOARDMAKER files and simply type 'BM' [Return] and the program will start. If you have any trouble installing or running BOARDMAKER and can't correct the problem, please call TSIEN and a technical support engineer will help you find the problem and determine a cure.

BOARDMAKER conventions

Certain keyboard and menu conventions are used in BOARDMAKER in an effort to accelerate an understanding of the facilities and use of the program. BOARDMAKER provides a number of ways to access these facilities using

either the cursor keys, a mouse or direct keystrokes which caters for both the occasional and experienced user.

Function keys

When a keyboard key is referred to in the manual, it is enclosed in square brackets. For example [Return] applies to the return key - [F2] to function key 2 and so on. When you are using a mouse and the manual or program refers to the [Return] key, then you can press either the return key or the left hand mouse button. Equally, when reference is made to the [Esc] key, you can press either the Esc key or the right hand mouse button. You will sometimes be asked to 'click' on a menu option. This simply means press and release the left hand mouse button when the cursor is on a menu option or move the cursor to the menu option and press [Return].

BOARDMAKER allows you to change the operation of the function keys for either the IBM-XT or IBM-AT keyboard layout. Consequential, the manual makes reference to both setups as follows : [F2]-(XT) or [F5]-(AT)

Shift, Alt and Control keys

Some keys are designed to be pressed simultaneously with other keys, in which case they will be written in the form [Shift][L] - meaning press the shift key and the L key at the same time. BOARDMAKER uses the [Shift], [Alt] and [Ctrl] keys in this way for various commands and functions.

When reference is made to a menu it is written in braces in the form {Config}{Block options}. This means click on the Config menu item and another menu will appear with one of the options as Block options. If you then click on Block options you will have selected the command referred to.

A comprehensive list of the equivalent menu and keystroke commands is given in an Appendix at the back of this manual.

Using Menus

In keeping with BOARDMAKER's sensible user interface all the menus which are presented to the user are hierarchical. This simply means that when a menu option is selected which presents a further menu, they build up on top of each

other in a sensible way - you can always see what route you took to reach the option you wanted and you won't be lost in a sea of pop up menus without knowing where you are.

Selection banner

Load	Alt+L
Save	Alt+S
Dir *.PCB	Alt+D
Symbol Library ...	
New	Alt+N
Output	...
Net	...
Exit editor	Alt+X

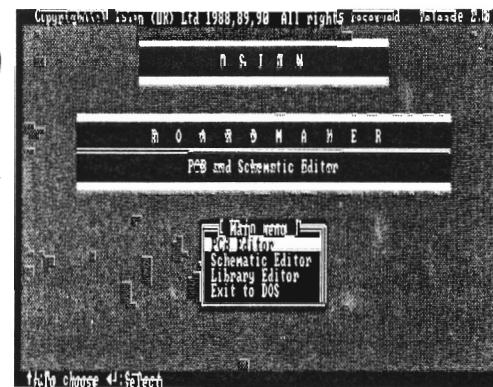
When a menu pops up, one of the options will always be highlighted. This is called the selection banner and it can be moved up and down using either the mouse or the cursor keys. You select the option by moving the banner to the desired option and pressing [Return] or clicking the left mouse button. If you want to drop or close the menu without selecting or changing

any option, simply press the [Esc] key or click the right mouse button.

When a menu is presented, if there are any special instructions for its use, they will generally be displayed along the bottom of the screen.

The BOARDMAKER main menu

When you start BOARDMAKER you will be presented with the main menu. This is the menu which allows you to select which of the graphical editors you want to use or Exit back to DOS.



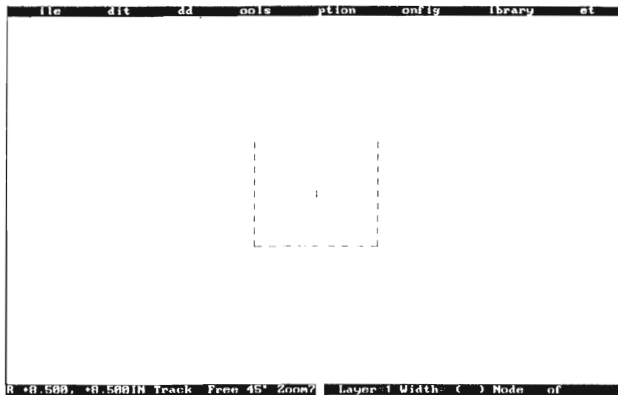
Whenever you have finished using the editor selected, you will be returned to this menu. The options available are :

- ☐ create or edit PCB layouts
- ☐ create or edit schematic diagrams
- ☐ create or edit symbol libraries
- ☐ exit back to DOS

To select the editor you require, move the banner using the mouse, cursor keys and press [Return] or press the first letter of the menu option.

The BOARDMAKER screen

BOARDMAKER's editor screens are identical whether you are using the PCB, schematic or library editor.



- ☐ the menu banner at the top of the screen which is the root of all menu choices
- ☐ the workspace or drawing area for your layouts and drawings indicated by the dotted box
- ☐ the status banner at the bottom left of the screen which gives details relevant to the screen status
- ☐ the information banner at the bottom right of the screen which gives information relevant to the current drawing mode.

Messages and line editor

BOARDMAKER will display any messages over the status banner for a pre-determined time which can be changed by the user. This area is also used for a line editor. When you need to enter a filename or library name, BOARDMAKER uses a simple line editor for entering text.

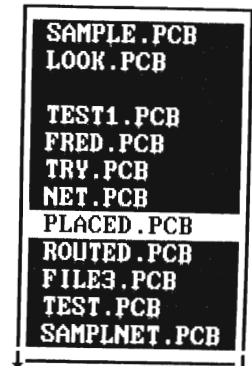
Editing text : Is there life on Europa_

The line editor is always in insert mode and responds to the following keys :

- ☐ Left arrow : moves the cursor to the left
- ☐ Right arrow : moves the cursor to the right
- ☐ [Del] : deletes characters at the cursor
- ☐ [Backspace] : deletes character to the left of the cursor
- ☐ [Ctrl][Del] : clears the line editor
- ☐ [Home] : moves the cursor to the start of the text
- ☐ [End] : moves the cursor to the end
- ☐ [Esc] : exit the line editor and cancel any changes
- ☐ [Return] : exit the line editor and accept the changes made.

When you are prompted for a file name or symbol name and want to see a list of the available choices, simply press [Return] when the line editor is blank and the {Option} menu will present a list of file names or symbol names. You can move the banner and select the item you want using the following keys

- ☐ Down arrow : moves the banner down one item
- ☐ Up arrow : moves the banner up one item
- ☐ [PgUp] : pages up
- ☐ [PgDn] : pages down
- ☐ [Home] : move to the first item
- ☐ [End] : move to the last item
- ☐ [Return] : selects the item
- ☐ [Esc] : closes the {Option} menu and cancels the operation



Currently loaded or selected items will be displayed in red.

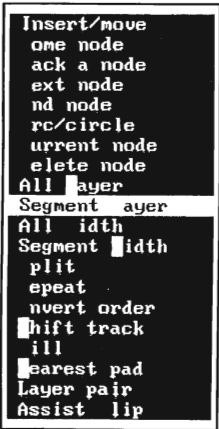
CGA and HGA display limitations

Due to the limitations of CGA, MCGA and HGA displays certain of the above screen details are not continuously displayed when using this type of display adaptor. The menu banner at the top of the screen will not show the {Library} or {Net} entry. These are accessible by selecting {Library..} or {Net..} from the {File} menu. The information banner is not shown. To see the mode information, you will need to press and hold the numeric keypad [+] key and it will be displayed over the status banner for as long as you keep the key pressed.

Menus or keystrokes ?

When you select an item from the menu banner at the top of the screen by clicking on it, the options which appear are chosen by then clicking on the menu option you want. The equivalent command using keystrokes are also displayed in the menus, either as function key numbers or highlighted letters in the menu option. Letters highlighted in a box require you to press the [Shift] key and the highlighted letter simultaneously.

The Option menu



All the menu banner options are available at all times with the exception of the {Option} item which is context sensitive. In other words, the options available at any time depends on the mode you have selected - this is the mode displayed in the middle of the status banner. If you select the {Option} menu and there is no mode selected, the menu will drop down and display {No Mode Selected}.

The menu shown here is an example of the option menu when you have selected Track mode.

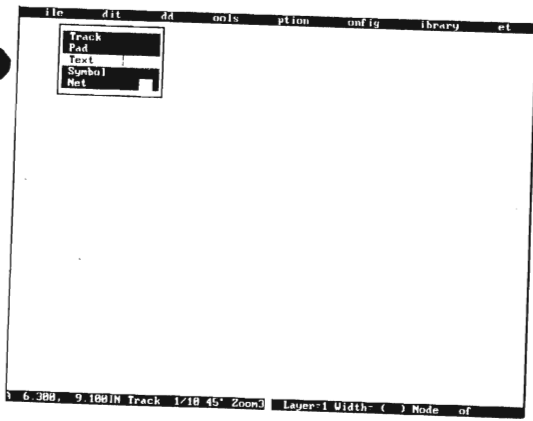
Chapter 3

User Guide

This user guide explains the interactive use of BOARDMAKER including sections dealing with the terms, concepts and ways to get the most out of the program. BOARDMAKER is designed to be :

- ☐ visual - Everything is shown on the screen as you would expect to see it using conventional tape-up methods. You simply make the screen look like the results you want to produce on the final artwork.
- ☐ intuitive - You are encouraged to explore the menus and options when trying out a new operation, your best guess is probably the correct one. Of course, the reference manual gives a full description of any commands you need.
- ☐ menu or key driven - You can use any combination of menus and keystrokes; start off by using the menus and as you learn the options, you can by-pass the menus by using the equivalent keystrokes.

A Quick Tour



Selecting a command is a simple matter of clicking on one of the items on the menu banner at the top of the screen or making selections by direct keystrokes. The menu banner can be considered as the root menu from which you can reach all the available submenus. Examples of the type of menus you will be using are shown over the page.

File menu

Load Alt+L
 Save Alt+S
 Dir *.PCB Alt+D
 Symbol Library ...
 New Alt+N
 Output ...
 Net ...
 Exit editor Alt+X

Add menu

Track
 Pad
 Text
 Symbol
 Net

Tools menu

Hilight F9
 Block F10
 oom
 nzoom
 an
 Set rigin
 Goto rigin
 Rubberband
 uasi-track
 Auto an
 rid on/off
 ngie assist
 rid snap
 Cursor Mode
 ursor type
 mits
 iew node

Edit menu

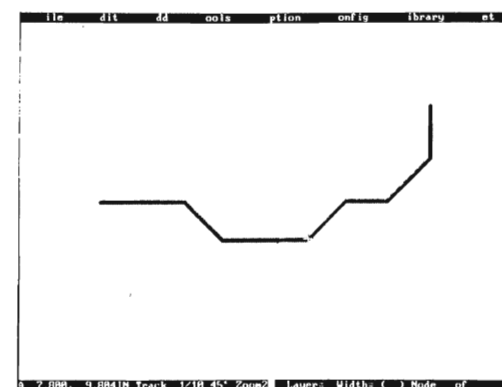
Track
 Pad
 Text
 Symbol
 Net

Library menu

View library
 Browse libraries
 Add to library
 Erase symbol
 Update symbols
 Export symbol

Net menu

Load nets
 Export nets
 Create nets
 Clear nets
 Edit net
 Show nets
 Hide nets
 Optimize
 Next net
 Back net
 Full DRC

The Anatomy of Tracks

A track can be thought of as a series of straight lines joined together at points or nodes. This would normally join two or more component pins together and may extend over a number of copper layers using via holes. As track segments are placed in sequence, their end points or nodes are 'fixed' down onto the drawing area and they build into a structure

which has a defined start point - the Home node, a number of intermediate nodes and a defined end point - the End node. This concept of the Home, End and intermediate nodes is important to understand when we come to edit existing tracks or track segments since we will need to move along the track from Home to End to select a particular node to move or delete.

Nets and netlists

Before you start designing a PCB layout, you need to know all the connections that must be made to ensure it is electrically correct. This information is, of course, implied in any circuit schematic you have, but for layout purposes, what you really need is a list of the pin numbers of components which are to be connected together. The following are typical examples :

Example 1

Connection list for signal called 4 MHz CLK might be

IC24 {14013}, 1 {CLOCK} to IC27 {14385}, 1 {CLOCK}

which means, connect IC24 {which is a 14013} pin 1 {which is called CLOCK} to IC27 {which is a 14385} pin 1 {which is called CLOCK}

Example 2

Connection list for signal called GROUND might be

TR1 {TO92}, EMITTER to R2 {AXIAL0.3}, pin 1

which means, connect TR1 {which is called EMITTER} to R2 {which is called AXIAL0.3} pin 1

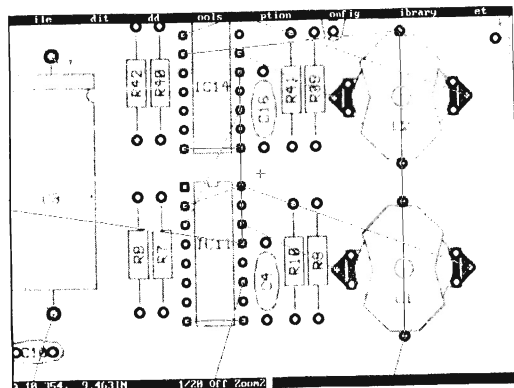
Each of these connections is called a **net** and the connections for any circuit can be fully described by a set of these - this is called a **netlist**.

In general terms, BOARDMAKER nets can be described using example 1 above :

- ☐ The name for the net is 4 MHz CLK
- ☐ IC24 - this is the component designator which would normally appear on your silk screen legend
- ☐ 14013 - this is the name given to the PCB layout symbol
- ☐ 1 - this is the pin number of the IC
- ☐ CLOCK - this is the Alias of the pin number i.e. another name by which the pin is known

Nets are drawn on the screen as a series of thin white lines which connect the nodes or component pins on the net together - the net nodes are drawn as thin

white diagonal crosses. This set of connections is sometimes referred to as a ratsnest and is, essentially, the information which is used to route and check the tracks. In a similar way to tracks, nets have a home node, end node and the intermediate nodes are numbered sequentially.



Design rules

When designing a PCB layout, there are certain rules of thumb which you probably apply when it comes to choosing the minimum track widths and the spacing between the tracks and pads. Design rules are a way of telling BOARDMAKER what these are - you specify the minimum track to track, track to pad and pad to pad separations you want to adhere to and the design rule checker will automatically warn you if you breach any of them.

The Current Mode

Whilst using BOARDMAKER you need to understand the concept of the current mode. At any particular time BOARDMAKER can be in one of the following modes :

- ☐ Track - for track operations
- ☐ Pad - for pad operations
- ☐ Text - for text operations
- ☐ Symbol - for symbol operations
- ☐ Block - for block operations
- ☐ Hilite - for highlight operations
- ☐ Desig - for designator operations in the PCB library editor
- ☐ Ref - for reference operations in the PCB library editor

This mode will be displayed in the status banner at the bottom of the screen and is selected by choosing the relevant item from the {Edit} or {Add} menus on the menu banner. Block mode is selected from the {Tools} menu or by using the direct key command [F10]. When a mode has been selected, the information banner at the bottom of the screen will display details relating to the current mode and the {Option} menu will be enabled so that you can use it to access the commands relevant to the mode.

The Current Item

This is the current item which is selected or 'noted'. This could be a track segment, a pad, a net, some text or a symbol. When you want to select an item,

either select the mode from the menu and then click on the item to select, or move the cursor over the item and press the direct command key.

Selecting tracks

When you are selecting a track, remember that you will need to select it near a node rather than in the middle of the track. The track segment selected will be drawn with the corners lowlighted and the current track is considered to be the one of which the current track segment is a part.

The Option Menu and Information Banner

The {Option} menu and information banner are enabled only when you are in a selected mode. If no mode is displayed on the status banner, then the information banner will be blank and the {Option} menu will do nothing when you select it. These two items and the mode selected, work together to give you all the information relevant to the mode you are in and access to pertinent commands. Some commands, such as the {File} and {Tools} commands are mode independent and are available at all times. The reference section gives details of the mode required for each command.

The following are examples of the details displayed on the information banner for each mode available :

Track Mode **Layer=1 Width= () Node of**

The 'M' indicates that the current track editing mode is Move node. This can be switched to Insert node by pressing the [Ins] key, in which case the display will show an 'I'.

The layer the current track segment is on or will be on if you are adding a new track, is copper layer 1. If you are using either the upper or lower silk screen layer then this will display '0S' or '9S' respectively. The width of the current track is width number 3 which has been set to an actual width of 19 thou. The last item shows that the current node is number 3 on a track made up of 12 nodes. i.e 11 track segments

Pad Mode **Layer= Width= () Hole size=**

This indicates that the pad is on all layers, although this could display any number between 1 and 8, the pad width number used is 11 which has been set to an actual outer diameter of 69 thou with a hole size of 27 thou.

Text Mode **Layer= Width= () Size= ()**

The current text is on the upper silk screen (layer 0), it uses text width number 1 which has an actual line width of 2 thou and uses text size number 6 which has an actual height of 200 thou.

Symbol Mode **(),**

The current symbol designator is R1 which is the symbol name AXIAL0.3 and has a value of 10K.

Block Mode **Track= Pad= Text= Symbol=**

This indicates that the block has been defined which has picked up 4 tracks, 16 pads, 2 text strings and 1 symbol which will be involved in a subsequent block operation.

Net mode **Net= Tot=**

This shows you that the current net is called N00006, there are a total of 33 nodes on the net and the design rules clearances for this net are : Track to track 10 thou, Track to pad 10 thou and Pad to pad 10 thou.

The Status Banner

The status banner displays information relating to the current BOARDMAKER status. It is shown at the bottom of the screen in the following format :

A 8.392, 10.592IN Track 1/40 45" Zoom3

The first character 'A' indicates that the current CURSOR MODE is absolute and is referenced to the fixed origin. This can be changed to be relative to a floating origin, in which case the status banner will display an 'R'.

The next two sets of numbers show the cursor position in X and Y - the 'IN' indicates that the cursor units are reading in inches. This can be changed so the cursor readout is in millimetres - the 'IN' will be replaced by 'MM'. Also, when the cursor readout is relative, the X and Y positions are prefixed by a '+' or '-' sign.

'Track' shows that the current mode is track mode. This area will display any one of the possible modes - Track, Pad, Text, Symbol, Block, Hilite, Net, Desig or Ref.

'1/40' is the current setting for the snap grid. BOARDMAKER provides six degrees of GRID SNAP in the range 1/10th to 1/80th. There is a further setting called 'Free' or free hand which gives you a nominal resolution of 2 thou.

'45' displays the current ANGLE ASSIST setting - this can be displayed as '45', '90', 'Off'.

'Zoom3' is the current level of ZOOM you have selected, in this case 3.

Layout, schematic or library editor ?

When you have just started BOARDMAKER, you are presented with a copyright message, release number and the main menu. The menu allows you to select one of four options :

- ☐ PCB Layout Editor - for designing your PCB layouts
- ☐ Schematic Editor - for designing your circuit diagrams
- ☐ Library Editor - to create or edit symbol libraries
- ☐ Exit to DOS - to finish using BOARDMAKER

The differences between the layout and schematic editor are minimal - layers have no significance for schematics, neither do the Gerber or NC drill outputs. The LAYER SELECTION window which allows you to select the layers in layout mode is changed so that you can choose the colours of the items drawn. Also, some options in the {Config} menu which are not relevant to schematics, do not appear. e.g. Resist mask etc.

The Library Editor is used when you want to create or modify schematic or layout symbols, although you can, if you wish, create a layout or schematic macro (a group of items) in this editor which can be used as a symbol.

How to get the most out of BOARDMAKER

BOARDMAKER has been designed to allow you to use it in a number of ways. If you want to start right out and design your layout manually by simply placing tracks and pads in a similar manner to traditional tape-up methods, there is nothing to stop you doing it. However, BOARDMAKER embodies a set of powerful tools and utilities which help ease you through the design path and produce a layout which is right first time. If you take the time to do a little preparation before you start - the routing of the board becomes remarkably straightforward and the checking when you have finished is done automatically for you - if you don't use netlists you cannot take advantage of these features.

Net list creation

Communicating with a schematic capture package

If you have a schematic capture package which you use to generate your circuit diagrams, the netlist it creates can probably be imported into BOARDMAKER. This is the most painless way of creating your layout since the netlist will be automatically generated and usually validated by the package. Although, BOARDMAKER has its own netlist format, there are integrated utilities which allow you to translate from one format to the other. Take the time to look at the READ.ME file on System disk 1 for the latest list of formats catered for.

The net import procedure is fully described under the heading Load Nets. However, in brief, BOARDMAKER will read, translate and validate a net list and allow you to quickly and easily map the schematic components described in it to PCB layout symbols. Extensive pin name validation is performed to reduce the possibility of errors. When the mapping is complete, BOARDMAKER will return you to the PCB editor and the symbols requested by the netlist are automatically placed complete with component designators

and ratsnest. All that is needed now, is for you to move the components to their final positions and route the board.

Creating you own net list

BOARDMAKER includes a powerful graphical net list editor. It allows you to create a net list quickly and easily and by taking the time to generate one before you start laying out your board, you can use the global design rule checker when you have finished the routing to automatically validate the layout - no more tedious manual verification of a check plot against a schematic.

To use the net editor, simply load up the all symbols you plan to use in your layout - don't worry about their actual position for now, this will be sorted out later. *As you place each symbol, give it a designator - as each symbol is added, BOARDMAKER will automatically increment the designators for you, but you may want to change them in line with your own component numbering.* When they are all placed, use the ADD NET and EDIT NET commands to create the nets which describe your circuit. Although this may seem a waste of time and perhaps an unnecessary complication before you start the actual layout, you will save considerable time when it comes to routing and the final checking of your board.

Now all you need to do is move the symbols into their approximate final positions on the board and start routing.

Placing the components

Probably the best way to make a start is to define the outline of your board somewhere in the middle of the available drawing area; this is best achieved by designing it as a symbol. There is **no need** to do your work close to the fixed origin down near the bottom left hand corner, you can use the floating origin to make the cursor readout relative to a datum on your board.

If you use the net optimize facility prior to moving the components you can tidy up the way the nets are drawn on the screen. When you edit a symbol, all nets will be hidden apart from those associated with the selected symbol. Since this effectively shows the density of connections to this component, it will help you to decide on the best place to put it.

Move the components one by one into their approximate final positions within the board outline you have drawn - *move the part in any direction, mirror it, rotate it and its connections will dynamically rubberband.*

Routing the board

Once all the components are placed, you are ready to start routing the board by placing tracks and vias to connect the components together. To aid this procedure, BOARDMAKER will selectively display the nets and keep account of which connections have been correctly routed.

Firstly, you will probably want to route the power or any other sensitive tracks. To do this, hide all the nets {Net}{Hide nets}{All nets} and then display the required net by name {Net}{Show net}{Net by name} - you will then see the ratsnest for the selected net and the net nodes are displayed as white diagonal crosses. By displaying the nets in this way, you don't have to continually refer back to your schematic to see what needs connecting together since it is displayed on the screen. When you have connected these points together with tracks and pads you need to verify the connection by highlighting the net {Tools}{Highlight} and performing a design rule check. These two functions will verify the route, issue any warnings if there are problems or if the route is correct, mark it as complete and hide the correctly routed sections.

When these nets have been routed you can then route the remainder of the board. In this case, you don't have to selectively show the nets by name, you can use the NEXT NET command which hides all nets except the next un-routed one. Again, route the net, highlight it and do a design rule check. If you continue this procedure you will eventually reach a point when you select NEXT NET and the message appears :

>All nets appear to have been routed.

This indicates that you have routed all the nets and the layout is probably finished. When you have reached this stage, you are ready to perform the final automatic checking of the board using the Full design rule checker.

Changing your design

As you are laying out the board, you may want to make changes to the circuit. If you want, you can simply add more symbols and components to reflect the changes required. However, it is good design practice to include these changes

on the netlist since they can then be correctly validated by the design rule checker. This system ensures you can keep account of any modifications you make to the board. If you decide to just add the extra symbols, tracks etc, the checking procedure will identify them as possible errors.

Which layers to use

The recommended layers working layers are as follows :

- ☐ Layer 0 - top silk screen
- ☐ Layer 1 - the component side of the board
- ☐ Layer 2 - intermediate layer 2
- ☐ Layer 3 - intermediate layer 3
- ☐ Layer 4 - intermediate layer 4
- ☐ Layer 5 - intermediate layer 5
- ☐ Layer 6 - intermediate layer 6
- ☐ Layer 7 - intermediate layer 7
- ☐ Layer 8 - the non-component side of the board
- ☐ Layer 9 - the bottom silk screen

The intermediate layers can be used for multi-layer boards or you could assign one of them for mechanical information or other miscellaneous uses.

Checking the entire board

Using the Full design rule checker (Full DRC)

When you have finished routing the board, even though you have may have been validating each net as it was routed, it is recommended that you use the Full design rule checker to automatically perform the final validation of your layout. This will generate an extensive report, which you can use as confirmation that all the routes are correctly connected and the design rules are not violated.

Running the Full DRC can greatly enhance your peace of mind.

Exporting a netlist

If you need to manually check your work, this can be achieved by *creating* a netlist from the layout and exporting it to a file (see Export Nets). This file can then be printed and manually cross referenced to your schematic. If you have a schematic capture package which allows it, you may be able to use this netlist to back modify your schematic diagram.

Reports

These routines are provided to allow you to obtain a printed copy of all the details of your board. The following data can be reported :

- ☐ Statistical data such as board size and item totals
- ☐ List of PCB components and their positions
- ☐ A bill of materials
- ☐ Full design rule checker findings

Board manufacture

When your layout is complete you need to consider the best way to go about turning it into a real PCB. BOARDMAKER provides various computer-aided-manufacture (CAM) outputs, each of which is designed to generate accurate output for board production. All these do not have to be sent to the output device directly, they can be re-directed to a named file and printed or plotted at a later date or using another computer.

Matrix printer plots

If you only have a dot-matrix printer available (either 9 or 24 pin) you can output your design on separate sheets and then photographically transfer them onto film suitable for prototyping. In general, the inherent lack of resolution of dot-matrix printers requires you to output at 2:1 or 4:1 and then photo-reduce the result - of course, the width of your printer will limit the size of board you

can make. Generally, you should consider this method for check plots to visually check your design.

Laser

The laser printer output can produce a very accurate 1:1 output. If you use film rather than paper in the printer, you can quickly create excellent quality master artwork which can be used directly for board manufacture. This method is excellent for producing in-house prototypes.

Penplot

This is capable of producing multi-colour check plots and high-quality final artwork. You have complete control over the plotter parameters and plot format. You will also find that many mechanical drawing, desktop publishing and even wordprocessing software packages allow you to import files in the standard plotting language HPGL. Consequently, quality PCB and schematic artwork can be incorporated directly into technical documentation.

Photoplot

This output generates files in Gerber format suitable for use by a photoplotter. Unless you are equipped with one of your own, you can use the many bureaux available to produce 1:1 artwork with a typical accuracy of 0.5 thou at reasonable costs. Also, many bureaux will accept files by modem - you can then get the artwork by return of post. Contact Tsien for details of typical prices and recommended bureaux.

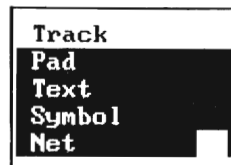
When this output is used in conjunction with the drilling output below, you will achieve the most accurate and professional results possible - this is the most highly recommended production path to use.

Numerically controlled (NC) drilling data

This is used to generate output files which dictate the drill hole co-ordinates and their sizes for use by your PCB manufacturer. This data is presented in Excellon ASCII format and is used to create the NC drilling tape - it saves you having to pay for the manufacturer to directly generate the tape from the artwork.

Placing Items and Associated Activities

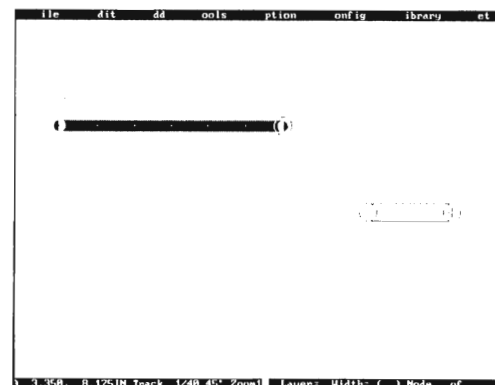
Placing tracks



This is used when you need to add a new track and can be performed using either the menus or the keyboard. From the menu select {Add}{Track} and BOARDMAKER will flash the message :

>Position cursor and press [Return]

You will now be in track mode. Move the cursor to the point you want the track to start and press [Return]. To add a new track using the keyboard, first move the cursor where you want the track to start and press [F2]-(XT) or [F5]-(AT).



As you now move the cursor you will see thin lines which show the outline of the track and possibly the design rule clearances if enabled. When the track is where you want it, press [Return] or the left hand mouse button and one track segment will be 'fixed' into place. If you again move the cursor to a new position, more segments will be added to the track - simply continue

moving the cursor and pressing [Return] until you have a complete track between the two points you want to join.

To finish placing the track, press [Esc] or the right hand mouse button. You can then move to another position to start laying another new track, but before you do that you might like to explore some of the other commands available in track mode.

When adding or editing tracks you have the option to change the way the track rubberbanding behaves. This Quasi-track mode is toggled on and off using the [Q] key or by selecting {Tools}{Quasi-track} and gives you a real-time display of the actual track width or the design rule checking information - you can then easily see whether a track will fit where you plan to position it. It can be setup to show either the true track width, the true DRC clearance or both.

To change the quasi-mode, select {Config}{Cursor settings} and you are presented with the {Quasi-track} and {Quasi-DRC} options. Simply click on these to view and select the desired setup. The Quasi-DRC can be selected to show either the track to track DRC or track to pad DRC.

Track width

BOARDMAKER provides you with eight different track width numbers, each of which can be assigned an actual width in the range 2 to 531 thou. This assignment is achieved using the {Track and pad sizes} option in the {Config} menu which is explained in the Changing Current Settings section. For now we will use the settings as they are to demonstrate the track facilities.

All width

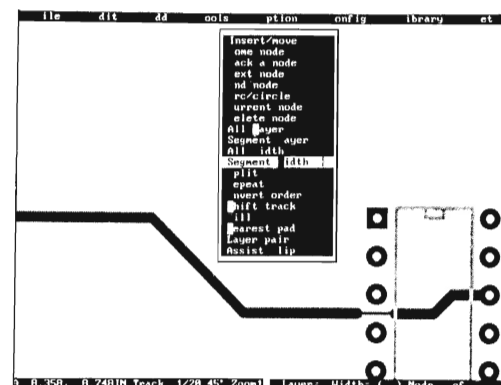
You can change the width of either the whole track you are adding or of the subsequent segments you are going to place. To change the width of the whole track use the ALL WIDTH command by pressing [W] or by selecting {Option}{All width}. You will be prompted to enter the width number for the whole track :

>Enter width number for the whole track

Enter a width number in the range 1 to 8 and all the track segments you have already placed and all subsequent tracks will be assigned the new track width. The information banner will display the current width number and the actual track width in thou in brackets.

Segment width

To leave the segments of track you have already placed and continue with a new width you need to use the SEGMENT WIDTH command. This is



particularly useful when you need to locally narrow a track to pass between two pads.

As you are placing a track, you select SEGMENT WIDTH by pressing [Shift][W] or selecting {Option}{Segment width} and BOARDMAKER will prompt

>Enter width number

When you enter a width number in the range 1 to 8 and continue to place track segments you will be using the new track width selected - the actual width of the current will be displayed on the information banner.

Changing layers

To change to a new layer is straight forward. You can choose to change the layer of all the track segments in the current track or change the layer of the current segment.

Segment layer

This allows you to switch to a new layer, leaving the segments you have already placed unaffected. If AUTO VIA has been enabled (see Current settings) then a via hole can be automatically placed. As you are placing a new track, select SEGMENT LAYER by pressing [L] or selecting {Option}{Segment layer} and you will be prompted :

>Enter the layer (0-9)

Select the new layer you want to use. If the layer is not enabled you will be warned and you will have to turn the layer on using the LAYER SELECTION command in {Config}{Layer selection}.

Vias, tracks and layer pairs

If AUTO VIA is ON you will then be prompted :

>Via hole ? [Return]-accept [Esc]-ignore

Press [Return] if you want to place a via hole, otherwise press [Esc]. You will then be able to continue tracking on the new layer in a new colour. The information banner will keep you up to date with the current layer you are on.

If AUTO VIA is set to NO CONFIRM then you can quickly switch to a new layer and place a via with any prompting, by pressing [Space]. This option is set up in the { Config} screen under the heading {Layer\Via setup}. You define the layer which you want to be paired together - typically for double sided layouts, layer 1 (component side) and layer 8 (non-component side), should be paired.

All layer

This command allows you to move all the track segments on the current track to one specified layer, regardless of the layer they may be currently on. While adding a track, select ALL LAYERS by pressing [Shift][L] or selecting {Option}{All layers} and you will be prompted :

>Enter layer for the whole track (0-9)

Enter the layer you want and all the track segments will be move to the selected layer and be drawn in the appropriate colour. If the layer selected was not turned on, you will need to turn it on using {Layer selection} or select another layer.

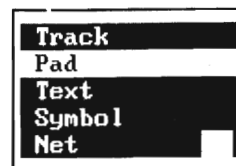
Angle assist

As you are moving the cursor in track mode you will see that BOARDMAKER will automatically introduce a 45 degree corner between adjacent track segments. This ANGLE ASSIST will give you a neat and ordered layout, but if you want to, it can be changed to give you 90 degree corners or indeed any angle by turning ANGLE ASSIST off. You can try changing these settings by pressing [Shift][A] or by using the {Tools}{Angle assist} menu option while you are adding a new track.

Assist flip

Another command worth experimenting with at this stage is the ASSIST FLIP. When you are adding a new track and are using the 45 or 90 ANGLE ASSIST, you may find that the corner which is automatically introduced is in the wrong place. This can be moved using the ASSIST FLIP command - {Option}{Assist flip} or [F].

Placing pads



To place a pad on the board, simply select {Add}{Pad} and you will be in pad mode. BOARDMAKER will give the message :

>Position cursor and press
[Return]

Move the cursor to the position where you want the pad and press [Return] or the left mouse button and the pad will appear. Although you can now see the pad it has not yet been fixed into place and is still mobile. If you move the cursor and again press [Return], the pad will move to the new cursor position. At this point, you can also change the layer, shape or width of the pad.

Pad layer

Pads are normally placed on all layers and are shown in white, however, you can choose a particular layer for the pad using the LAYER command. Press [L] or select {Option}{Layer} and BOARDMAKER will give the message :

>Please enter 1-8 or A for all layers

When you enter the layer number, the pad will only appear on the layer selected.

Pad shape

To change the shape of the pad you are placing use the SHAPE command by selecting {Option}{Shape} or pressing [S] and the {Option} menu will present the 14 available shapes



BOARDMAKER has to offer. To select the one you want, simply click on the menu item and the pad you have placed will change to the new shape.

Pad width

Using the WIDTH command, you can choose a new inside and outside diameter for the pad. BOARDMAKER allows you to choose from 16 different pad sizes - each of these are referred to as width numbers and can be assigned a wide selection of actual diameters using the {Track and pad sizes} menu. This is explained under the heading Current Settings. For now we will use the sizes as they are currently set. Select WIDTH by pressing [W] or selecting {Option}{Width} and BOARDMAKER will prompt you with :

>Enter width number (1-16)

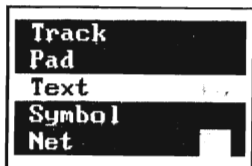
When you have entered the number followed by [Return], the pad will take on the attributes for that width number and will be displayed with the new outer and inner diameter.

All the above changes to the pad will be updated on the information banner so you can immediately see the actual settings for the pad. Whenever you are prompted to enter a setting, you always have the option to cancel by pressing [Esc] or the right mouse button.

Fixing the pad

When you are happy with the position, shape, width and layer of the pad, you can fix it by pressing the [Esc] key or the right mouse button. To place another pad, move the cursor to a new position and press [Return] - a new pad will then be displayed *with the same attributes as the last pad placed*.

Placing text



To place text on the drawing, select {Add}{Text} or press [F6]-(XT) or [F7]-(AT) and you will be in text mode. If you selected {Add}{Text}, BOARDMAKER will give the message :

>Position cursor and press [Return]

Move the cursor to the position where you want the text and press [Return] or the left mouse button and you will be prompted to enter some text.

>Editing text :

Enter the text you want followed by [Return] and the text will be displayed on the screen. Although you can now see the text it has not yet been fixed into place and is still mobile. If you move the cursor and again press [Return], it will move to the new cursor position. At this point, you can also change the size, width or layer of the text before you actually fix it into place.

Text size

BOARDMAKER provides you with eight sizes of text. To change the size of the text you have just placed use the SIZE command by pressing [S] or selecting {Option}{Size} and you will prompted to enter a new text size.

>Please enter text size number (1-8)

For details of the actual text sizes see SIZE in the command reference. Enter the size you want and the text will be displayed in the new size.

Text width

Text can be drawn using one of four line widths. This is chosen using the WIDTH command. Press [W] or select {Option}{Width} and BOARDMAKER will prompt you :

>Please enter width number (1-4)

For details of the actual text line widths see WIDTH in the command reference. Enter the width you want and the text will be displayed using the new line width.

Text layer

Text can be placed on any layer which is turned on. To change the layer the text is currently on, use the LAYER command. Press [L] or select {Option}{Layer} and BOARDMAKER will give the message :

>Enter the layer (0-9)

User Guide

When you enter the layer number, the text will move to the layer you selected.

All the above changes to the text will be updated on the information banner so you can immediately see the settings selected. Whenever you are prompted to enter a setting, you always have the option to cancel by pressing [Esc] or the right mouse button.

Incrementing and decrementing text

When drawing schematics you often need to increment or decrement numbers and letters, for example on connectors or IC pin numbers. These two commands allow you to easily do this. When you have placed the first text item, say 'IC2a', move the cursor to the next position and repeat it by pressing [R]. Increment it by pressing [I] and the new text will be 'IC2b'. This can be used in many ways and greatly speeds up the process of numbering components.

Fixing the text

When you are happy with the position, size, height and layer of the text, you can fix it in position by pressing the [Esc] key or the right mouse button.

Placing components or symbols

BOARDMAKER allows you to place pre-defined symbols which were created in the Symbol Library editor. These symbols, which at their simplest level are single component or schematic outlines, are stored in a named library. It is possible to have many libraries, perhaps grouping similar types of components or arranged in a project based manner. Different libraries exist for layout and schematic symbols - the default library filenames are SYMBOL.LL for layout symbols and SYMBOL.SL for schematic symbols.

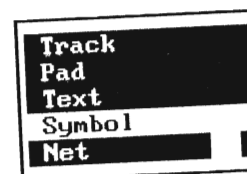
When you are laying out a PCB or drawing a schematic, you can of course draw individual component outlines using combinations of tracks and pads. These could then be copied using a block command. However, this is very wasteful of the system memory in your computer. Using symbols is by far the best way to place components; it is a simple procedure, it makes best use of the available memory and the screen re-draw time is faster.

More importantly, if you use symbols which have been designed with designators and references (more about these later), you can take full advantage of the net facilities and automatic checking procedures built into BOARDMAKER. You will also find the component numbering a breeze, since they will automatically increment as you place the symbols.

Symbols vs Macros

In the BOARDMAKER library editor you can arrange any combination of tracks, pads or text to form a symbol. Although, you will probably find that the majority of your layout symbols will consist of pads and a silk screen outline, you can also create a complete layout which includes tracks. These layout symbols or macros, are particularly useful in situations where you want to reuse a group of components over a range of PCBs or indeed repeat them on the same PCB.

Placing a symbol



To place a symbol on the board, simply select {Add}{Symbol} or [F8] and you will be in symbol mode. BOARDMAKER will give the message :

>Position cursor and press [Return]

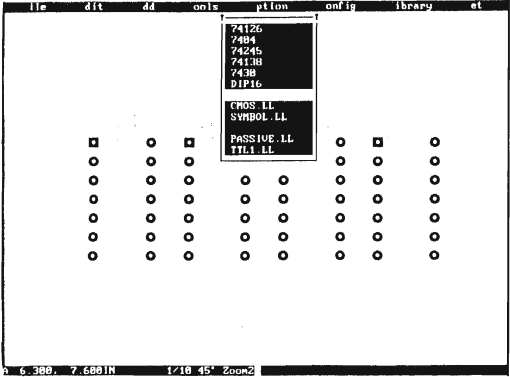
Move the cursor to the position where you want the symbol and press [Return] or the left mouse button

and you will be prompted for the symbol name :

>Enter symbol name :

Enter the name of the symbol you want to place, e.g. 'DIP14' [Return] or if you cannot remember the names, press [Return] on its own and the option menu will present you with a list of the symbols in the current library. Any symbols which are already in the drawing will be displayed in red and any other libraries are shown at the bottom of the menu. If you want to examine the other libraries contents, simply select it and the menu will then display its contents.

When you press [Return] to choose the symbol, the screen will clear (*you will not loose any work !*) and the symbol will be shown graphically. At this point



you can either place it in the drawing by pressing [Return] or flip through the other symbols by pressing [N] for the next symbol or [B] to go back a symbol.

When selected, the symbol will appear in the drawing area. Although it is now displayed it has not yet been fixed into place and is still

mobile. If you move the cursor and again press [Return], it will move to the new cursor position. At this point, you could rotate or mirror the symbol into the exact position you require.

If the symbol was designed with a designator e.g. IC, this will automatically be incremented and displayed on one of the silk screen layers.

Fixing the symbol

When you are happy with the position of the symbol, you can fix it in position by pressing the [Esc] key or the right mouse button.

Placing circles and arcs

You may find certain instances when you need to use circles and arcs in your layouts or schematics. Some RF applications, for example, require round rather than angular corners for tracks, or perhaps more commonly, silk screen component outlines for radial capacitors. BOARDMAKER constructs circles by expanding a single track segment with exactly two nodes into a circle. The home node defines the centre of the circle and the end node defines a point on the circumference.

To create a circle, select {Add}{Track}, move the cursor to the position you want the centre to be and press [Return]. Now move the cursor a position where you want the circle to pass through and press [Return]. You have now defined a track segment which is the radius of the circle. Select {Option}{Arc/circle} and you will be prompted :

>Tolerance (Low/Med/High/[Return]=auto) ?



By pressing [L], [M], [H] or [Return] you can select the number of track segments which are used to construct the circle - auto, will select a value for you based on the size of the circle. A circle will then be drawn on the screen made up of short track segments.

Creating an arc

To create an arc, you simply delete the track segments from the circle just drawn. Select delete [D] and the segments will be deleted from the end node. Repeat this until you have the arc you require. It can then be rotated, mirrored or repeated as required.

Press [Esc] when you are happy with the circle or arc you have created, this will fix it in position.

Editing items

The Edit facility provides a method for making changes to tracks, pads, text or symbols that you have already placed on a layout or schematic.

Editing a track or track segment

Once a track has been placed you can, at any time, go back and modify it. This can be applied to the whole track or just to individual track segments. Once a track is selected for editing, the option menu will present all the available track editing options as shown here.

Selecting the track

Before you can edit the track you will first need to select or note it. To do this, select {Edit}{Track} and you will be prompted :

>Position cursor and press [Return]

Move the cursor to a point near a NODE on the track and press [Return]. The track segment is now selected and is drawn with the ends lowlighted. You can now move back and forward along the track using BACK [B], NEXT [N], HOME [H] and END[E].

Changing layers

To change the layer of the selected track segment use SEGMENT LAYER [L] from the {Option} menu, if you want to change the layer for the whole track use ALL LAYER from the same menu. Simply follow the prompts presented to do either of these functions.

Changing track widths

You can change the width for the track segment selected or for the whole track using SEGMENT WIDTH [Shift][W] or ALL WIDTH [W] respectively. To change the width of an individual track segment, first select the track the segment is on, use {Back a node} or {Next node} to select the segment you want and press [Shift][W] to select SEGMENT WIDTH. You then enter the width you want the segment to be.

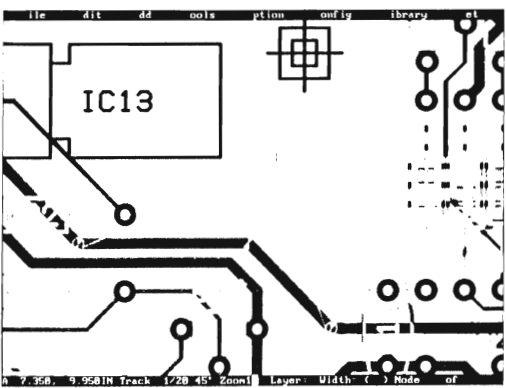
Moving a complete track

To move the whole track to a new position, simply select the track, move the cursor to where you want the home node of the track to be and select SHIFT by pressing [Shift][S].

Splitting a track

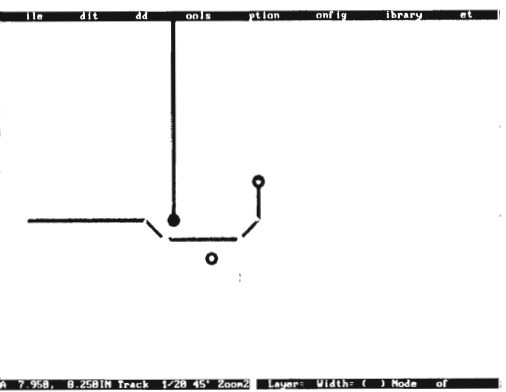
Using {Option}{Split track} allows you to break a track into two sections - it breaks the track from the current node.

Moving track segments



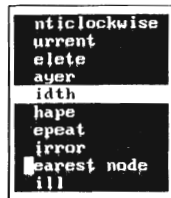
BOARDMAKER allows you to move tracks by moving the current node to a new position dragging the two track segments joined to it. First, select the track you want to move as detailed earlier, then select {Option}{Insert/move} or press the [Ins] key until an 'M' appears in the information banner. This indicates that you are in move rather than insert mode. Now move the cursor and you will be able to re-position the node and associated tracks to a new position using the guide lines drawn. When you are ready, press [Return] to actually move the node and track segments. Press [Esc] to fix the track(s) in the new position.

Inserting track segments



This option allows you to insert new track segments from anywhere along an existing track. Repeat the above procedure, but this time press the [Ins] key until an 'I', for insert, appears in the information banner. Now move the cursor and press [Return] and a new track segment will be inserted. Repeat this until you have inserted as many segments as you need then press [Esc] to finish.

Editing pads



When you need to change the layer, width or shape of an existing pad then choose the {Edit}{Pad} option. The commands to do these operations are identical to those you used when you added the pad and are displayed in the pad option menu shown here. You will be prompted to :

>Move cursor and press [Return]

Move the cursor to the pad you want to edit and press [Return].

Changing layer, width or shape

You can now change the layer using the LAYER [L] command, the pad width using the WIDTH [W] command or the pad shape using the SHAPE [S] command. Note that the width refers to a width number which defines the inner and outer diameter of the pad.

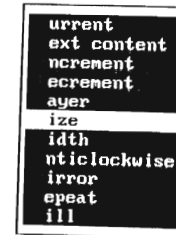
A further command you may like to look at is MIRROR [M] - this is useful for surface mount pads and not only mirrors the pad in a vertical axis, but also logically inverts the layer it is on, so swapping it from one side of the board to the other. These commands are also available in the {Options} menu when the pad has been selected. Follow the prompts for each command or use the Command Reference for exact details of their functionality.

Moving the pad

To move the pad to a new position, simply select it, move the cursor to the new position and press [Return]. Although, the pad will move to the new position, it is still mobile - just move the cursor again and press [Return] to re-position it. When you are happy with the position, press [Esc] to fix it in position.

Editing text

BOARDMAKER allows you to change the layer, width or size of any existing text on a drawing using the same commands you used when you originally



placed the text. Further to this, you can rotate or mirror it or if you need to, change its textual content. The option menu will present all the text editing options available to you as shown here.

Selecting the text

Before you can edit the text you will first need to select or note it. To do this, select {Edit}{Text} and you will be prompted :

>Position cursor and press [Return]

Move the cursor to a point near the start of the text and press [Return]. The text is now selected and is drawn with the corners lowlighted.

Changing layer, width and size

These are changed using the LAYER [L], WIDTH [W] or SIZE [S] commands and are identical to those functions you used when you added the text. These are explained in the section Placing text and are fully detailed in the Command Reference.

Rotating text

When the text has been selected, you can rotate it anticlockwise in steps of 90 degrees using the ANTICLOCKWISE command. This allows you to put it in one of four positions rotated about the centre of the first character of the text string. This can be achieved using {Option}{Anticlockwise} or by pressing the [A] key - the text will rotate through 90 degrees each time you select the command.

Mirroring text

BOARDMAKER allows you to mirror any text you have placed on the screen. It mirrors it in a vertical axis through the centre of the first character of the text string and also inverts the layer the text is on. If the text was on layer 0, the top silk screen, for example, it will be swapped to layer 9, the bottom silk screen. This effectively swaps the text from one side of the board to the other, whilst retaining its 'readability' from that side of the board. This is useful, for example,

when you need to switch a surface mount legend from one side of the board to the other.

Changing the textual content

This allows you change the content of the text you have previously placed. Select the text and then use the {Option}{Text content} command and you will be prompted :

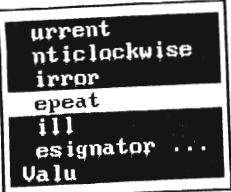
>Editing text : This is some text

Where 'This is some text' is the text you are editing. You can now use the line editing facilities to change the text. When you have finished, press [Return] and the changed text will be displayed on the layout or schematic - if you want to abandon the changes then press [Esc].

Moving the text

To move the text to a new position, simply select it, move the cursor to the new position and press [Return]. Although, the text will move to the new position, it is still mobile - just move the cursor again and press [Return] to re-position it. When you finished, press [Esc] to fix it and drop out of text mode

Editing a symbol



When you want to perform an operation on an existing symbol you first need to select it. To do this, select {Edit}{Symbol} and you will be prompted :

>Position cursor and press [Return]

Move the cursor to a point near the origin of a symbol and press [Return]. The symbol is now selected and is drawn with the lines lowlighted. The options available

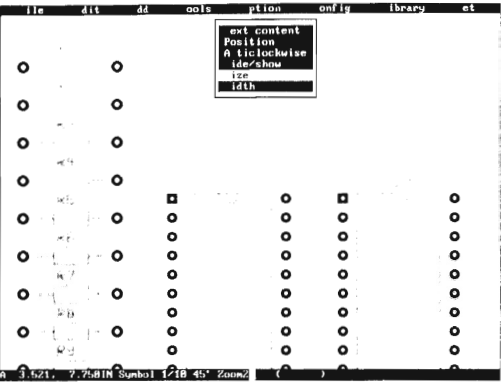
to you are displayed in the option menu shown here.

Moving a symbol

To move the symbol to a new position, simply select it, move the cursor to the new position and press [Return]. Although, it will move to the new position, it is still mobile - just move the cursor again and press [Return] to re-position it.

When you are happy with the position, press [Esc] to fix it. As the symbol is moved, its designator will move with it.

Changing the designator



To add or edit the designator, select {Option}{Designator ...} or press [D] and the {Option} menu will present the possible designator options.

Select {Text content} and you are prompted for the designator :

>Editing designator

Enter the new designator and press [Return] - you will be warned if it is not unique.

Moving the designator

When you place a symbol, the designator will appear in a default position. If you want to move it, edit the symbol and select {Option}{Designator ...}. Choose {Position} from the {Option} menu which appears and you will be prompted to move the cursor to the new designator position and press [Return].

A quick way to move the designator is by using the direct key strokes. When the symbol is selected, you can move it by positioning the cursor and pressing [Return]. To move the designator on its own, simply press and hold the [Shift] key and again put the cursor in the new position and press [Return] or the left hand mouse button.

Rotating the designator

The designator can be rotated independently of the symbol by selecting {Option}{Designator ..} and selecting {Anticlockwise} from the {Option} menu or press [N]. The designator will be rotated 90 degrees anticlockwise.

Hiding and showing the designator

If you do not want a designator to appear on your silk screen, you can choose to hide or show it using {Option}{Designator...}{Hide/show} to toggle the text between hidden and shown.

Changing the designator size and width

When you place a symbol, the designator text will default to the size used when it was designed. If this is not the right size you can use {Option}{Designator...}{Size} or [S] to change the height of the text to any one of the eight sizes available. Equally, you can change the text width by selecting {Option}{Designator...}{Width} or [W].

Giving a component a value

As well as being able to give a symbol a component number, you can also give it a value. If you have imported a net from a schematic capture package then this information will already be loaded. However, if you want to change it or add a value to a symbol, select {Option}{Value} and you are prompted :

>Editing value :

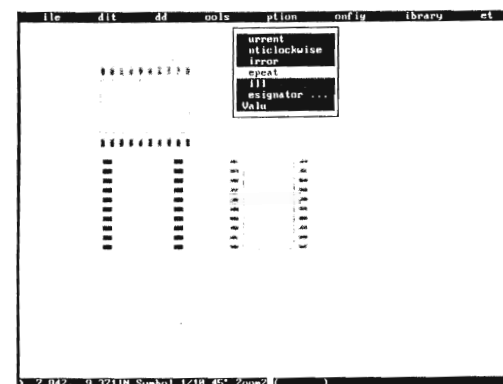
Enter the value required and press [Return].

Rotating a symbol

You can rotate the selected symbol anticlockwise in increments of 90 degrees using the ANTICLOCKWISE command. Select {Option}{Anticlockwise} and the symbol will be rotated 90 degrees anticlockwise about the symbol origin. Repeat this until it is in the position you want and press [Esc] to fix it. Note the orientation of the designator is maintained.

Mirroring a symbol

This command primarily caters for surface mount devices. It allows you to mirror the current symbol about a vertical axis through its origin and logically invert all the layers of the symbol. This means that a symbol comprising a set



of pads on layer 1 only and a silk screen outline on layer 0 - the upper silk screen, will, when mirrored, put the pads on layer 8 and the silk screen outline on layer 9 - the bottom silk screen. Any text that was part of the symbol will also be mirrored as will any tracks. This facility allows you to easily swap a surface mount component from one side of the board to the other. You

don't have to define a symbol for each side of the board - one symbol can be rotated and mirrored to cope with any placement requirement.

Deleting items

BOARDMAKER allows you delete either whole tracks, track segments, pads, text or symbols. In some instances when you have deleted an item, you may see some residue on the display, if you re-draw by selecting PAN [P], these will disappear. In general the DELETE command applies only to deleting track segments and the KILL command applies to deleting whole items such as tracks, pads, text or symbols.

Deleting tracks

You can delete either a single track segment or a whole track using DELETE [D] or KILL [K] respectively. You must first select the track or segment you want to delete by selecting {Edit}{Track}, moving to a point near a node on the track and pressing [Return]. Use the HOME, END, BACK and NEXT commands to select the exact segment you want to work on.

Delete a track segment

You can delete a track segment from the middle or from the end of a track using the DELETE [D] command by selecting {Option}{Delete}. When you delete from the end node, you are effectively 'ripping' up the track from the

end to the home node. If you want to delete the other way, from the home to the end node, you may first need to invert the order of the track nodes using the INVERT ORDER [I] command - select {Option}{Invert order}.

When you delete a track segment from the middle of a track, the current node will be removed and the two track segments joined to it will be replaced by a new track segment between the previous and next node. You may find that the RUBBERBAND lines which appear when you are deleting these middle segments confusing, if so then disable it by pressing [Shift][B] or selecting {Tools}{Rubberband}.

Delete a whole track

If you have selected a track segment, you can delete the whole track connected to it using the KILL [K] command. You will be prompted :

>Please confirm. Type (Y)es or (N)o.

Press [Y] to delete the track. If you are sure of what you are doing, you can delete without the confirmation using [Shift][K].

Deleting pads

Before you delete a pad, you have to select it using {Edit}{Pad}, move the cursor to the pad you want to delete and press [Return]. Select {Option}{Kill} and you will be prompted :

>Please confirm. Type (Y)es or (N)o.

Press [Y] to delete the pad.

Don't forget, you cannot directly delete pads which are part of a symbol, you have to delete the whole symbol or edit it using the library editor.

Deleting text

Before you delete text, you have to select it using {Edit}{Text}, move the cursor to the text you want to delete and press [Return]. Select {Option}{Kill} and you will be prompted :

>Please confirm. Type (Y)es or (N)o.

Press [Y] to delete the text.

Deleting a symbol

Before you delete a symbol, you have to select it using {Edit}{Symbol}, move the cursor to the symbol you want to delete and press [Return]. Select {Option}{Kill} and you will be prompted :

>Please confirm. Type (Y)es or (N)o.

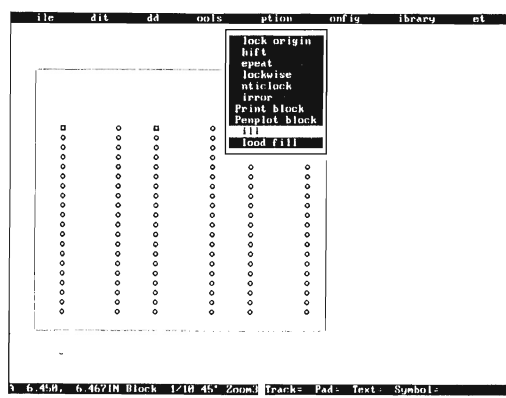
Press [Y] to delete the symbol.

Block manipulation

The block functions are the most powerful manipulation commands in BOARDMAKER. You are able to put a block around a group of items on the screen, define which type of item is going to be affected by the next block command, then repeat, delete, rotate, mirror or move those items in the block. Where tracks are concerned, you can choose whether the connectivity is maintained or not using the Rubberbanding facility in the {Config}{Block settings} menu.

Defining the block

Before you can perform any of the block operations, you need to draw a block around the items you want to manipulate. You do this by selecting block mode, press [F10] or select {Tools}{Block}.



Drawing the block

Put the cursor where you want the block to start, press [F10] to select block mode and move the cursor. You will now see the block box with the first corner at the cursor position when you pressed [F10] and the other corner which can be dragged

around with the cursor. If the first corner is not in the position you want, press [Esc] to drop the block, move the cursor to the required position and press [Return] to start again.

Drag the block box until it encloses the items you want to operate on and press [Return]. The block is now defined and its origin is set to the first corner you placed.

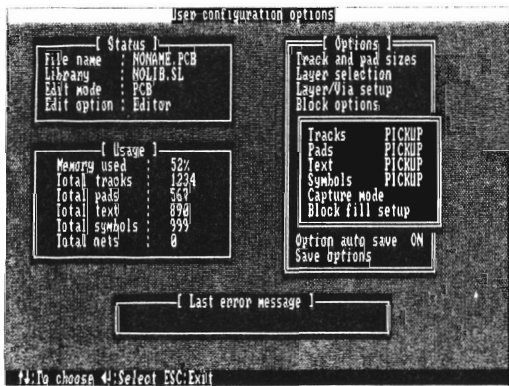
Note that only items wholly enclosed by the block will be affected, with the exception of tracks with {Capture mode} is set to Rubberbanding on (more of this later). Symbols, text and pads will only be selected if their origin lies within the block.

Changing the block origin

You may find you need to change the block origin to something other than the corner of the block. You can do this using BLOCK ORIGIN - {Option}{Block origin} or press [B]. This is convenient when you are moving a block and you want to position an item in the block at a specific point. In this case set the origin to the position of the item.

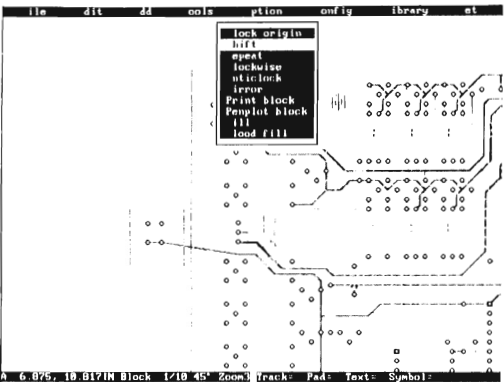
Defining which items are affected

When you perform a block operation you define which items are picked up by the block. For example, you may want to delete all the pads in a block, but leave the tracks, text and symbols unaffected. You choose the items affected from the {Config} screen under {Block options}. Select {Config}{Block options} and you will be presented with a menu displaying the current block settings. You can choose whether to PICKUP or LEAVE tracks, pads, text or symbols by clicking on the relevant item.



Note that you can only change these when a block has not been defined.

Maintaining connectivity



The last item in this menu is the capture mode. This can be set to rubberband tracks which pass over the block boundary by turning Rubberbanding on. Consequently, any subsequent block operations will maintain track connectivity as a 'rats nest'.

Deleting a block

To delete a block of items, first define the items you want to delete using {Config}{Block options} - you MUST set the {Capture mode} to {Rubberbanding off} and then define the block. BOARDMAKER will not let you delete a block if this is set on.

To delete the items in the block select {Option}{Kill} or KILL [K] and you will be prompted:

>Please confirm. Type (Y)es or (N)o.

Press [Y] to confirm or [N] if you decide against it. A word of caution - block delete is a powerful command, you could delete a whole layout or schematic if you are not careful! If are worried about what you are about to do, make an extra backup of the file you are working on - just in case !!!

Moving a block

To move a block of items, again select those items you want the block to pickup or leave using {Config}{Block options} and then define the block. To move the block, position the cursor where you want the origin of the block to be moved to (use the BLOCK ORIGIN command to set it), select

{Option}{Shift} and the block will move to the new position. If you set the {Capture mode} to Rubberbanding on, then the tracks will rubberband to the new position. Press [Esc] to drop the block.

Repeating a block

This copy facility is almost identical to the SHIFT command described above, except that the selected contents of the block are copied to the new position rather than just moved. Also, the {Capture mode} MUST be set to Rubberbanding off BEFORE you define the block.

Once the block is defined, select {Option}{Repeat}, move the cursor where you want the origin of the copied block to be and press [Return]. Alternatively, position the cursor first and press [R] for REPEAT. The selected contents will then be copied to the new position.

If you there were symbols with designators in the block move, the new symbols created will also have suitably incremented designators.

Rotating a block

BOARDMAKER allows you to rotate a block clockwise or anticlockwise in increments of 90 degrees **about the cursor position**. If you set the {Block options} to pickup all items, you can use these commands to rotate the whole board - useful if you are trying to output a layout which is too wide to fit your printer or plotter.

Clockwise

To move a rotate a group of items clockwise, select those items you want the block to pickup or leave using {Config}{Block options} and then define the block. To rotate the block, position the cursor where you want the block to be rotated about, select {Option}{Clockwise} and the block will rotate clockwise by 90 degrees. Repeat this until the block is in the position you want. If you set the {Capture mode} to Rubberbanding on, then the tracks will rubberband to the new position.

Anticlockwise

This is identical to the Clockwise command except (surprise, surprise), the block is rotated anticlockwise. Select {Option}{Anticlockwise} or use the direct command key [A] once the block has been defined.

Mirroring a block

This command is designed as a tool for manipulating surface mount devices or mirroring a board prior to plotting. It allows you to mirror the selected components in the block about a vertical line through the cursor and logically invert all the layers. This means that when you have put a block around a group of surface mount items and associated tracks you can then swap it to the other side of the board. A block comprising a set of pads on layer 1 only and a silk screen outline on layer 0 - the upper silk screen, will, when mirrored, put the pads on layer 8 and the silk screen outline on layer 9 - the bottom silk screen. Any text that is in the block will be mirrored and so will any tracks.

This is a powerful command with many implications - you can, if you want, mirror the whole board about the cursor position. Be sure of what you are doing before you actually do it, and if necessary, make a backup of the drawing before you start.

To mirror a group of items, define those items you want the block to pickup or leave using {Config}{Block option} and then define the block. To mirror the block, position the cursor where you want the block to be mirrored about, select {Option}{Mirror} and the block will be mirrored. If you set the {Capture mode} to Rubberbanding on, then the tracks will rubberband to the new position and switch to their logically inverted layer - you may need to add new via holes in some cases.

Printing or plotting a block

You may find you need to print or plot a small section of your layout or schematic. You can achieve this by first defining a block and then selecting the {Option} menu. Choose either {Print block} or {Penplot block} and you will be presented with the printer or plotter setup screens. For details of how to actually output from these screens see the Output section in this chapter or in the Command reference

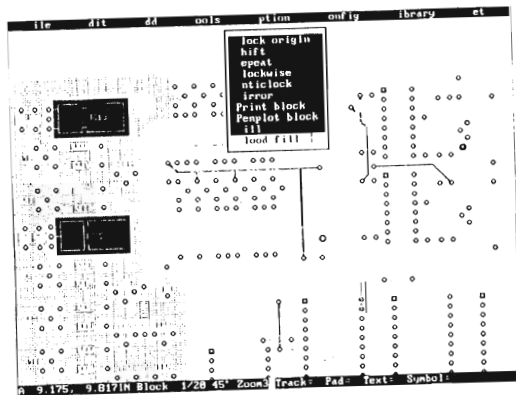
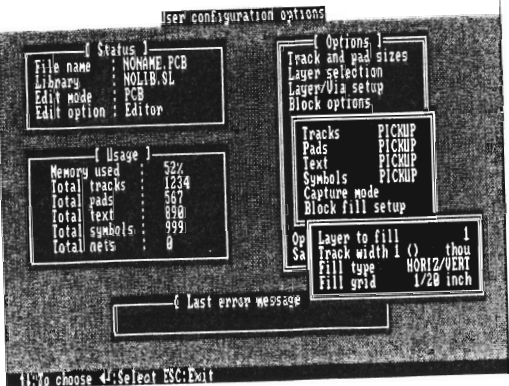
Creating a ground plane

This facility allows you to define and fill an area with copper automatically avoiding any tracks and pads which are already placed using the design rules set. It has been implemented as an option in block mode and the fill attributes can be configured as required. Select { C o n f i g } { B l o c k options } { Block fill setup } and you will be presented with the setup menu to change the track width, pitch or grid, the layer you want to fill and the fill style. *The design rules are used to determine the clearances around any tracks and pads. It is good practice to set the global design rules equal to the largest of the net design rules.*

Once the block fill settings have been made you can then proceed with the actual filling of the block.

Put a block around the area you want fill by selecting { Tools } { Block } or by pressing [F10] and dragging the block box to cover the required area - press [Return] to complete the box. To fill the block, select { Option } { Flood fill } and you are prompted :

>Position cursor and press [Return]

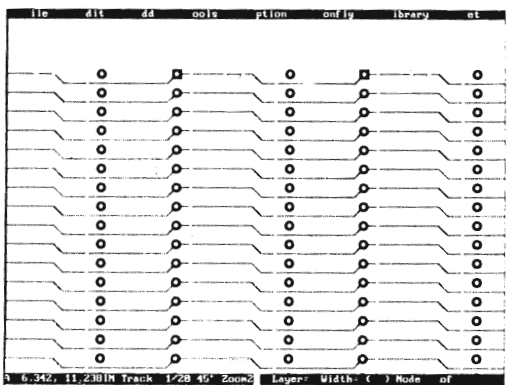


Move the cursor to a vacant point within the block and press [Return] - this will be used as the starting or seed point for filling. Ideally, choose the largest space in the block which does not contain a track or pad.

BOARDMAKER will then create a filled area within the block which is connected to the start or seed point. If there are vacant areas in the block which cannot be connected because there are no routes to them, they will not be filled.

Repeating items

BOARDMAKER provides a repeat command for all modes of operation including block mode as described above. When using this command, you are effectively borrowing the attributes of an existing item on the layout or schematic and using them to create a new item.



Repeating tracks

This facility is useful when you want to generate the tracks for address and data bus placement. To do this, place the first track of the bus and turn the rubberbanding off by pressing [Shift][B]. Select { Option } { Repeat }, move the cursor to the position you want the new track's home

node to be and press [Return]. The track will then be copied to the new position. If it is not in the right place, use { Option } { Shift } or press [S]. To create more tracks, simply repeat the above procedure for each one.

Repeating pads

Select the pad on the drawing you want to copy, select { Option } { Repeat }, move the cursor to the position you want the new pad to be and press [Return].

A new pad will then be drawn on the screen with the same size, shape and layer as the original. BOARDMAKER will warn you if you attempt to repeat the pad onto itself.

Repeating text

Using this command, you can copy existing text to a new position on your layout or schematic. Select the text you want to copy, select {Option}{Repeat}, move the cursor to the position you want the new text to be and press [Return]. New text will then be drawn on the screen with the same content, size, width and layer as the original. BOARDMAKER will warn you if you attempt to repeat the text onto itself.

Repeating symbols

Using this command, you can copy an existing symbol to a new position on your layout or schematic. Select the symbol you want to copy, select {Option}{Repeat}, move the cursor to the position you want the new symbol to be and press [Return]. A new symbol will then be drawn on the screen exactly like the original but in the new position. BOARDMAKER will warn you if you attempt to repeat the symbol onto itself. If the symbol had a designator, then this will be automatically incremented for the repeated symbol.

Current settings

To view or change the current settings you need to use the {Config} menu, although some settings are available from other menus.

Changing the track and pad sizes

BOARDMAKER allows you to have eight different track widths and sixteen different pad widths on any one drawing. Each of these widths can be set in the range 2 to 531 thou. The track and pad widths particular to that drawing will automatically be loaded with the drawing. Changing the width of track number 3 to 6 thou, will change every track on the layout which used track width 3 to 6 thou. This provides a powerful method of making global changes to the drawing - for example to increase the size of all tracks of width 10 thou up to 19 thou,

simply change the width of the track number set to 10 thou to 19 thou and all the track widths will be increased. This can of course be applied to lines in schematics and pad inner and outer diameters.

One of the pads, width number 16, is reserved for auto-via holes. When a via is placed using the Auto via facility, this is the size of pad that is used.

When you are looking at the track and pad sizes display, the widths which are currently in use are shown in red text and the unused sizes are shown in black text. This immediately tells you which track and pad sizes you have used on the drawing.

To select the Track and pad sizes screen select, choose {Config}{Track and pad sizes}. It presents a table of the track and pads available and allows you to change their sizes. Move the menu banner over the size you want to change using the mouse or the arrow keys and use the [+] and [-] key to either increase or decrease the size selected. You can also do this by holding the left hand mouse button down and moving the mouse forward to increase the size and backwards to decrease the size.

Layer and colour selection

By selecting {Config}{Layer selection} you will be presented with a table showing which layers are enabled and what colour has been assigned to each layer. To change the colour, simply move the cursor to the colour you want the layer to be and press [Return] or the left mouse button. The check marks will show what colour each layer is. To disable a layer, move the check mark to black.

For schematics, this window does not show the layer number, since they have no significance for schematics, instead, you select the colour you want the lines, text and symbols to be drawn in.

The range of colours available will change according to the display adaptor you are using. BOARDMAKER auto-detects the adaptor and changes the available colours. For CGA displays you have the choice of three colours plus black - for MCGA, EGA or VGA displays, you have the choice of six colours plus black.

Fixed and floating origins

BOARDMAKER provides two cursor modes, one of which is absolute, the other is relative. If the CURSOR MODE is set to absolute, then the cursor readouts on the status banner are the X and Y distances from the bottom left hand corner of the available drawing area, the fixed origin. If the CURSOR MODE is relative, the cursor readouts are the X and Y offsets from a floating origin set by the user. The status banner always displays the current CURSOR MODE as an 'A' for absolute or an 'R' for relative - select {Tools}{Cursor mode} or press [M].

SET ORIGIN [O] allows you to set the floating origin to a position to any position on the board. {Tools}{Goto origin} will move the cursor to the floating origin you have set. The default floating origin is the same as the fixed origin.

Cursor settings

As well as changing the CURSOR MODE to be relative or absolute, you can also change the type of cursor using {Tools}{Cursor type}, to switch between a full cross-hair and a target cursor. The full cross-hair is useful for aligning adjacent items. You can also change the size of the target cursor using CURSOR SIZE. Select {Config}{Cursor size} and you are able to choose between a small, medium or large target cursor.

Auto vias

BOARDMAKER can be set to automatically place a via hole when you are placing a track and switch layers. By switching {Config}{Layer/via setup}{Auto via} to ON, you will be prompted when you change layers:

>via hole ? [Return] - accept [Esc] - ignore

Press [Return] if you want a via hole, otherwise press [Esc]

Pad size number 16 is reserved for auto-vias and can be changed to suit your needs.

If the {Auto via} is set to NO CONFIRM, it is placed automatically without confirmation.

Layer pairs

This is a facility which enables you to associate layers to ease the tracking process. When you press the [Space] key it will switch between the layers which are paired. If the Auto via option is set to NO CONFIRM, it allows you to quickly swap to the other side of the board and automatically place a via hole.

For example, with layers 1 and 8 paired, if you are routing on layer 1 and press [Space] you will change to layer 8 and vice versa.

Any combination of layer pairs can be set to suit you requirements.

Inches or millimetres ?

Most PCB layouts are usually designed using an imperial pitch. However, some component packages are arranged with a pin out on a metric pitch. This is provided for by allowing the cursor readout to be in inches or millimetres. Using this facility in conjunction with the GRID SNAP set to free hand, you can very accurately position items with a resolution of approximately 2 thou. You can define a symbol with a metric pin out which can then be called back later in the PCB editor over an imperial grid. You switch between the two types of units by selecting {Tools}{Units} or by pressing [Shift][U].

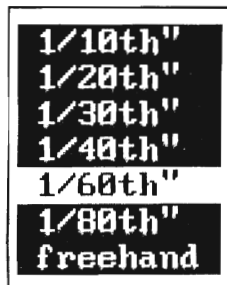
If you change the units, you will not affect any components which have already been placed.

Grid sizes and settings

Visible grid

BOARDMAKER provides two types of component placement grid. The first is a hidden grid that the cursor snaps to as it is moved around the screen and the second is a visible grid of dots which acts as a guide and reference on the screen. Selecting {Tools}{Grid on/off} will turn the visible grid on and off or use the direct command GRID [Shift][G].

Snap grid



BOARDMAKER provides six degrees of GRID SNAP in the range 1/10th inch to 1/80th inch - a further option is to select free hand which provides an approximate resolution of 2 thou. To choose a grid snap setting, select {Tools}{Grid snap} and the {Options} menu will present the settings available - click on the one you want and the new snap grid will be set - its value is displayed on the status banner.

Nearest pad and node snap

This option in Track or Pad mode allows you to position an item based on the position of other items on the board. When in track mode, {Option}{Nearest pad} or [Shift][N] will snap you to the nearest pad or symbol pad. When in pad mode it will snap you to the nearest track node. This is particularly useful when items on the layout have been placed with a grid snap setting which was different from the current setting. e.g metric components

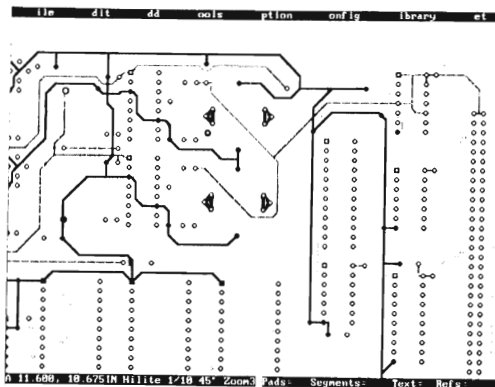
Net highlighting

This command allows you to examine the items which are connected on a layout and to verify that they are connected to the items you expected them to be. This facility has been implemented as a functional mode - like track, pad, symbol, text or block. The word 'Hilite' will appear in the status banner when the mode is active.

Highlighting a connection

When you are in the PCB editor, with your layout loaded, you can highlight a net by the menu selection {Tools}{Highlight} or by pressing the function key [F9]. Move the cursor over the item you want to highlight and press [F9], the status banner will display the progress of the highlighter - typically 'Highlighting

item 3 of 25', indicating that item 3 from a total of 25 items found connected, is being highlighted.



The items which are connected will now be displayed in white and the information banner gives you details of the items on the net. For example it could show 'Pads = 67 Segments = 168 Text = 2', which lets you know that there are 67 pads on the net, 168 track segments and 2 text characters.

The highlighter will display not only the connected items on the same layer as the first item you selected, but also any items which are connected on other layers by vias or pads even if these layers are disabled.

When the net has been highlighted, you are free to pan and zoom around the net for checking purposes. If you want to highlight another net, simply move the cursor to a new item to highlight and press [Return] - the new net will then be highlighted and the old one cleared.

You can interrupt the highlighting process at any time or clear a highlighted net by pressing [Esc].

Highlighter warnings

If you have selected the highlight mode by pressing [F9] or selecting {Tools}{Highlight} and the cursor has not been positioned over an item, you will be warned:

>Nothing to highlight at this point

In which case, move the cursor to a point where there is an item and try again.

It is possible that you can select a point where there are multiple nets. For example, if you select a track which is on layer 1 and there is another un-related track directly below it on layer 8, you will be warned :

>Multiple net. Please try another point

In this case, move the cursor to a unique point on the net you want and press [F9] again.

Deleting a highlighted net

Highlighted nets can be deleted using {Option}{Kill highlighted}. It will delete all highlighted tracks and pads which are not part of symbols. This function is useful when you need to delete an area which has been block filled.

Using and managing netlists

BOARDMAKER has been designed to be flexible enough to allow its many powerful features to be used in almost any combination. Many other PCB layout programs force you to use the net facilities - BOARDMAKER, however, lets you decide whether you want to use them. Here are four suggested ways in which you can use it :

Mode 1 - Using a schematic capture program

- ☐ Create the net list using a schematic capture program (OrCAD, Schema)
- ☐ Load the netlist into BOARDMAKER
- ☐ Translate the netlist to BOARDMAKER format
- ☐ Map the component names to specific PCB layout symbols
- ☐ Save the symbol map
- ☐ Automatically load and place the components complete with designators and values.
- ☐ Position the components using the net connection density as a guide to the best position.

- ☐ Use the Net editor to assign different design rules to the various nets - they all default to 10 thou.
- ☐ Route the board using the nets to show you which pins need to be connected and validate each route as you go using the Design Rule Checker (DRC)
- ☐ Use the Global DRC to check the entire board and print out a full report of its findings, including design rule violations, suspicious or extraneous connections and maximum and minimum track widths.
- ☐ Export the netlist, convert it into the required format and where possible, use it to back modify the schematic diagram.

Mode 2 - Using the built-in net list editor

- ☐ Load all the symbols you need into the PCB layout editor
- ☐ Create a net list by clicking on the points you want to be connected together.
- ☐ Position the components using the net connection density as a guide to the best position.
- ☐ Use the Net editor to assign different design rules to the various nets - they will default to 10 thou.
- ☐ Route the board using the nets to show you which pins need to be connected and validate each route as you go using the Design Rule Checker (DRC)
- ☐ Use the Global DRC to check the entire board and print out a full report of its findings, including design rule violations, suspicious or extraneous connections and maximum and minimum track widths.

Mode 3 - On the fly design with global rule checking

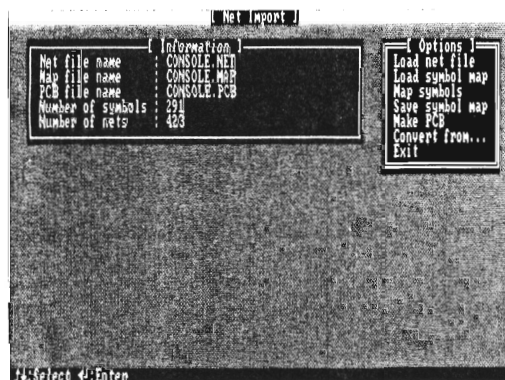
- ☐ Layout and route the board by placing symbols, tracks, pads and text
- ☐ Use the Design Rule Checker to check the clearances
- ☐ Use {Create nets} to make a netlist from the work you have done
- ☐ Export the net list to a file, obtain a printed copy and use it to check against your schematic diagram.

Mode 4 - On the fly design with manual checking

- ☐ Layout and route the board by placing symbols, tracks, pads and text
- ☐ Use the Design Rule Checker to check the clearances and manually check the connections you have made

If you do not have a schematic capture program, it is clear from these suggestions that Mode 2 is the most efficient way to design your layout. By taking time out to do a little preparation, the routing of the board is very straight forward and you can rely on BOARDMAKER to do all the final checking of the connections for you.

Loading a netlist



Select {Net}{Load nets} and you will be presented with the Import net screen. If the netlist you want to load is not in BOARDMAKER format you will first need to translate it. Select {Convert from ..} and choose the format required. You are then prompted to enter the name of your netlist file and the name for the new file to be created in

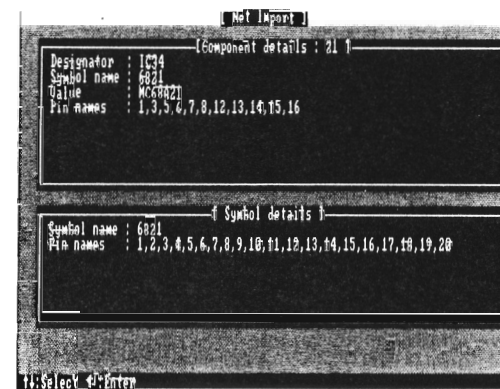
BOARDMAKER format. When the names have been entered the netlist will be translated.

To load the file, select {Load net file} and you are prompted for a name.

If you cannot remember the name of the file or simply want to see what files are available, press [Return] and a directory window will present a list. Put the banner over the name you want and press [Return] to select it.

Once the net file is successfully loaded, you need to map the component names to the PCB layout symbols you want to use. e.g. a component called 74LS00 could be mapped to a symbol called DIP14.

Select {Map symbols} and you will be presented with two windows.



The top window gives details about the component in the netlist, giving its designator, symbol name, value and pin names used. The bottom window gives details of the PCB layout symbol you want to map it to. You can use [PgUp], [PgDn], [Home] and [End] to flip through each of the components in the net list.

To map the symbol, press [Return] and you will be prompted for library name. Enter this directly or press [Return] to see a directory. When you have selected a library name, the window will show a list of the symbols in the library.

Put the cursor over the required symbol and press [Return]. Provided the pin names of the component are a subset of the pin names on the chosen symbol, the mapping is valid. You will be warned if there are discrepancies.

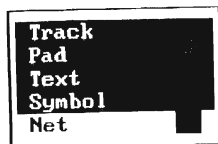
Continue this with each of the components until they are all successfully mapped. You can save the mapping at any time by pressing [Esc] and selecting {Save symbol map} and entering a name for the map file. Likewise, you can restore the to the state you left the mapping in by selecting {Load symbol map}.

When all symbols are mapped, select {Make PCB} and enter the name you want to give to your board - the components will then be automatically placed into the PCB editor. Choose {Exit} to return to the editor and start routing the board.

Using the netlist editor

BOARDMAKER has an integrated graphical net list editor which allows you to easily and quickly create nets before and during routing of the board. Initially, you need to load the symbols you want in your layout and then you simply click on the points you want to be on the net.

Adding a net



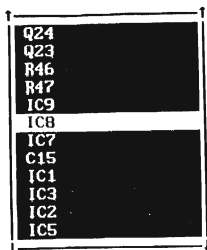
This is used to create a new net. You can use the menus or the keyboard. From the menu select {Add}{Net} and BOARDMAKER will display the message :

>Position cursor and press [Return]

You will now be in net mode. Move the cursor to the connection you want to be on the net and press [Return]. To add a new net using the keyboard, first move the cursor where you want the track to start and press [Shift][F2]-(XT) or [Shift][F5]-(AT). You are then prompted :

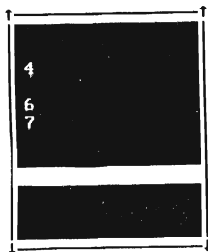
>Editing net name :

If possible, use a name which has relevance e.g. +12V, AD0 etc. If there is no suitable name, use the default convention N00001, N00002 and so on - BOARDMAKER will not let you enter duplicate net names and will automatically increment the default name. Press [Return] when ready.



To build up the net, you can simply move the cursor to each point you want to be on the net and press [Return]. Once again, BOARDMAKER can ease you through this process by presenting you with details of the components and their pins. Once you have started a new net, select {Option}{Add to by name} and you are prompted :

>Enter Designator, Pin name :

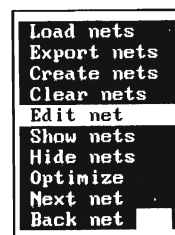


You can now enter the next point you want to be on the net in the form IC10,4 [Return] and this node will be added to the net - the screen will pan to the point if necessary. If simply press [Return] in reply to the prompt, the {Option} menu will present a list of all the component designators available. Put the banner over the one you require, press [Return] and you then get a list of all the pin names on that component.

This procedure is designed to be intuitive so that you can simply enter a designator name at the prompt and the {Option} menu will show you a list of the pins on the it.

Small diagonal white crosses will appear at the net nodes and they will be joined together by white lines which indicate the nets. When you have finished, press [Esc] or the right hand mouse button. You can then move to another point to start another net. Continue this procedure to quickly build up a netlist for the entire layout.

Editing a net



This command provides a method of editing nets which have already been placed on the layout. Since nets have a name, there are two ways of defining which net you want to edit - you can specify the net by node or by name.

If you want to edit a net and you know its name but are unsure of its exact nodes, select {Net}{Edit net} and the {Option} menu will present you with the names of all the nets. When you select the one you need to edit, all other nets will be hidden and the selected net will be displayed.

If you want to edit a net and you know where one of its nodes is, simply select {Edit}{Net} and you are prompted :

>Position cursor and press [Return]

Put the cursor over the desired net and press [Return] to select it.



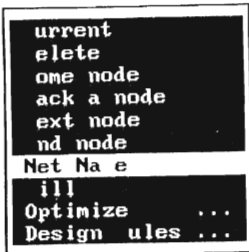
Moving along the net

In a similar manner to tracks, the net nodes are sequentially ordered. You can move back and forward along the track using {Back a node}, {Next node}, {Home node} and {End node}. These are available from the {Option} menu when the net has been selected for editing. change is selected.

Deleting nets and net nodes

Nets can be completely removed by selecting {Option}{Kill} or you can selectively delete net nodes by moving to the relevant node using {Back a node} or {Next node} and selecting {Delete} from the option menu.

Changing the net name

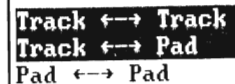


When the required net has been selected by editing it, you have the option to change its name. Select {Option}{Net name} or use the direct keystroke [M] and you will be prompted to enter the new name for the net. Any alpha numeric name can be entered up to 16 characters long. However, the name must be unique and BOARDMAKER will warn you if the name entered already exists.

Changing the design rules for a net

This is one of the powerful features of BOARDMAKER. It allows you to define not only a set of clearances or design rules which apply to the whole layout but also the design rules which are valid for each net. When you import a net or use the net editor to create a net, each one is given a set of default design rules. The ability to define different design rules for different nets ensures that BOARDMAKER can perform a very accurate design rule check. For example, you can have one set of design rules for power tracks and another for signals. Indeed, it is possible to give each net a completely different set of rules if you want.

To change the design rules, first edit the net and select {Option}{Design rules...} and the {Option} menu will present these options :

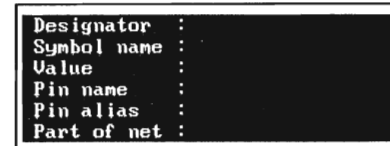


Put the cursor over the design rule you want to change and press [Return]. You are then typically prompted :

>Pad <---> Pad clearance is : 12 thou

To change this use the up or down arrow keys to increase or decrease it. Press [Return] when you have finished.

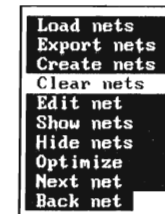
Viewing nets and nodes



When a symbols are loaded, you can view the details about each one very easily using the View node function. Select {Tools}{View node} or use the direct keystroke [V] and the {Option} menu will present details of the symbol pin nearest to the cursor.

When using the direct keystroke, the information will appear for as long as the key is held.

Clearing a netlist



If you want to clear an existing netlist from the current PCB, choose {Net}{Clear nets}. You are asked to confirm this operation :

>All nets will be lost. Proceed(Y/N)?

Choose [Y] to confirm the action and all the nets will be cleared.

Create nets

You can use this facility to create a netlist from the PCB you have designed. You may want to do this if you did not use the net list editor to create a netlist before you started laying out the board. The results can then be exported and a printed copy used to manually check the connections.

```

Load nets
Export nets
Create nets
Clear nets
Edit net
Show nets
Hide nets
Optimize
Next net
Back net

```

Choose {Net}{Create nets} and if there are any existing nets you will be prompted :

>All nets will be lost. Proceed (Y/N)?

Select [Y] and all the nets are highlighted one by one as the netlist is created.

Exporting the nets

All net information is held in the PCB file, consequently, if you want to see these you need to export it from the PCB file to a named net file. Select {Net}{Export nets} and you will be presented with the Export net screen.



The information window shows the name of the current PCB file and the net file name defaults to the same name, but with the file extension .NET. To create the net file, choose {Export net} and you are then prompted for the net file name.

Press [Return] to select the default name or enter new file name and the net file will be created.

The {Convert to ..} option allows you translate a BOARDMAKER net list to other net list formats. Select the format required and you are prompted for the BOARDMAKER net file name and the other format file name. When both are entered, the file will be translated to the desired format. Note that the original net file and PCB file will not be affected by this procedure.

Select {Exit} to return to the PCB editor.

Hiding and showing nets

You can use {Net}{Show nets} and {Net}{Hide nets} to show and hide selected nets or all nets. If the {All nets} option is selected, all the nets will be displayed as a ratsnest. If you want to show a selected net, then choose {Net by name} and select the net name required or choose {Net by node} to select the net associated with a particular node or {Net by component} to display the nets connected to a particular component.

```

Nets on component
Net by node
Net by name
All nets

```

Note that when you show nets, BOARDMAKER will not display nets which have been correctly routed and verified using the design rule checker. Since you are usually only concerned with those nets which are yet un-routed, BOARDMAKER will not bother you with them. If however, you need to see these nets, you need to optimize them.

Optimizing nets

When you have created or imported a net, the order of the net nodes may not be defined. Optimizing the nets re-orders the nodes by determining the shortest path between them. As well as tidying the way the nets are drawn, it allows you to see the next nearest point which needs to be connected while you are routing board. To do this choose {Net}{Optimize}. You are then prompted to choose the nets you want to optimize either {All nets} or by {Net by name}. When you have made your choice, you then specify shortest path, X-bias or Y-bias which allow you to bias the connection paths either horizontally or vertically.

```

Shortest
X Bias
Y Bias

```

Design rule checking

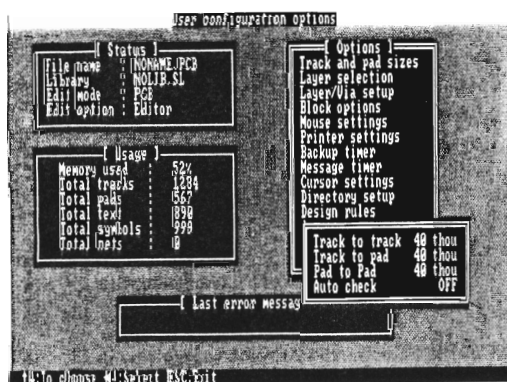
This option provides a method of checking to see that tracks and pads have been placed in accordance with a set of pre-defined rules or clearances. Consequently, this facility is split into two parts. The first, to set the design rules and the second, to apply the rules to a particular net.

The global design rules are those which will be applied for a DRC when there are no net based design rules. If you want to change the rules for a particular net, see the section under the heading Using and managing netlists.

The rule checker will also verify any associated nets or sub-nets against the connections it has found during the net highlight. This procedure ensures you only see the un-routed nets when you select NEXT or BACK NET.

Setting the global design rules

You set the global design rules from the Config screen by selecting {Config}{Design rules} and you will be presented with the design rule check (DRC) menu. As well as allowing you choose which rule you want to set, it displays the current clearances. You have the option to set the track to track, track to pad or pad to pad minimum clearances. The last option on the menu allows you to determine whether the design rule checker is automatically started when a net has been highlighted - by default, this is turned off.



Select the rule you want to set and you will be presented with the edit window which allows you to change the setting. Use either the up or down arrow keys to increase or decrease the displayed value - if you have a mouse, you can do this by moving the mouse forwards or backwards respectively.

Applying the design rules

To apply the design rules to a net, you must have first highlighted the net. If you have enabled auto check in {Config}{Design rules}, this will automatically take place after the net highlighting has finished. The progress of the DRC is displayed on the status banner in the following format :

>Using design rules for net N00001

where N00001 is the name of the net being checked or if there are no symbols with references it will display :

>Using global design rules for DRC

If there are some DRC violations, the information banner will display it in the following format :

T<-->T:12 T<-->P:23 P<-->P:6

This tells you there are 12 track to track violations, 23 track to pad violations and 6 pad to pad violations. If there was any text on the net, you will be warned:

>Text on net! Remove before proceeding

For DRC purposes, text is not permitted on a net.

Viewing the violations

The position of these violations are automatically stored so you can examine them. You use the [<] and [>] keys to view the previous or next violation. These can also be selected from the {Option} menu if required. Once you have checked these violations, you would typically want to edit out any problems. To do this, you simply select the edit mode you want, either track, pad, text or symbol and correct the offending item.

The [<] and [>] keys are still active even when the hilite mode has been cancelled. When there are current violations as a result of a DRC, the {Tools} menu has the options {Prev violation} and {Next violation} which duplicate the key functions [<] and [>]. You also have the option to clear the list of violations by selecting {Tools}{Clear violations}.

When you are viewing the violations using the {Next violation} and {Prev violation} options, the cursor will move to the relevant point, panning if necessary, and the status banner will display the message

>violation 2 of 34 : Track<-->track

indicating this is violation 2 from a list of 34 and is a track to track violation.

Using the Full design rule checker

This is used to apply the design rules, whether net based or global to the entire board. It is intended to perform the final check on your layout prior to committing it to manufacture. The process involves two parts - running the Full DRC to do a board-wide check including design rules, net violations, finding erroneous nets and unconnected pins - then generating an output report which can be printed.

This whole procedure may take several minutes for a complex design, but it is more efficient and accurate than manual methods and by using {Output}{Reports} a printed log of its findings along with an error/warning summary for you to examine.

These errors can also be shown on the screen by selecting {Tools}{Next violation} and {Tools}{Previous violation}. The error details will be displayed in the status banner.

Moving around your drawing

Pan and zoom facilities are provided to allow you to view or edit different parts of your drawing at whatever level of magnification is required. You will probably find you use these two functions more than any others which is why BOARDMAKER has been designed to re-draw the screen very quickly, even on the slower machines.

Zoom and Unzoom

BOARDMAKER has 7 levels of zoom, with Zoom level 1 being the highest magnification and Zoom 7 the lowest. If you need to look at the whole drawing then select Zoom level 7; you will see the greatest detail by selecting Zoom level 1. The current zoom setting is always displayed on the status banner at the bottom of the screen and if you select the zoom level already displayed the screen will redraw and pan so that the current cursor position becomes the centre of the screen.

Although you are able to select ZOOM by pressing [Z] or selecting {Tools}{Zoom}, BOARDMAKER recognizes the number keys at the top of the keyboard (not the numeric key pad) as the zoom level you require. In other words, pressing key [5] immediately selects zoom level 5 and so on.

Pan

When PAN is selected using {Tools}{Pan}, the current cursor position becomes the centre of the screen, so to pan to a particular area on the drawing, simply move the cursor to the area you want to see and select PAN [P].

If you have told BOARDMAKER you are using a three button mouse using the {Mouse type} option in the {Config} menu, then pressing the middle button on the mouse will also allow you to pan.

You can also automatically pan by half a screen when the cursor touches the edge of the screen. This feature can be toggled on and off from the {Tools}{Auto pan} menu or by the direct keystroke [Shift][P].

Tagging

This provides a convenient way of jumping from one part of the drawing to another. You simply place a tag at the point you want to return to frequently by positioning the cursor and pressing [Ctrl][1]. The number '1' will appear in a small box on the display to mark this point. To return to this tag, simply press [Alt][1] and you will be immediately panned back to this point. You can have up to nine tags per drawing using the numbers 1 to 9.

Drawing information

You can get information about your current drawing at any time from the Status and Usage windows on the {Config} screen.

Status window

The status window displays the current drawing and library filename - without the pathname, the current library name, the edit mode and edit option. The

edit mode is displays whether you have selected PCB or schematic mode, the edit option displays either editor or library editor - whichever you selected from the main menu.

Usage window

This displays details of how many tracks, pads, text and symbols are on the drawing. It also shows as a percentage, how much of the system memory you have used.

Managing your PCB and schematic files

The {File} menu contains all the commands you will need to load and save your drawings, as well as various other commands to see what files are on your disc. The last option in this menu allows you to exit the editor and return to the main menu. BOARDMAKER filenames conform to the standard DOS restriction of eight characters for the name and three for the extension. In order to differentiate between PCB and schematic files, the file extensions are fixed to .PCB and .SCH respectively.

Loading and saving files

These commands allow you to load or save PCB or schematic files using {File}{Load} or {File}{Save}. You will then be prompted for the name of the file you wish to load or save. If you cannot remember the name of the file you want to load, simply press [Return] on its own and the {Option} menu will present a list for you to choose from. When you enter the filename you do not need to enter the filename extension .PCB or .SCH, since this is already known by virtue of the edit mode you chose from the main opening menu. If you have made any changes to the drawing, you will be prompted to save it prior to loading a new one.

Load	Alt+L
Save	Alt+S
Dir *.PCB	Alt+D
Symbol Library ...	
New	Alt+N
Output	...
Net	...
Exit editor	Alt+X

Directory

If you want to see the names of the files on your disk, then you can view their names using the DIRECTORY command by selecting {File}{Dir *.PCB*.SCH}. The type of file names displayed will depend on the edit mode you are in. i.e. it only displays PCB files when you are in the PCB layout editor.

Press [Print screen] if you want a printed copy.

Clearing the drawing area

Use the NEW command to clear the workspace ready to start working on a new drawing. When you select {File}{New}, you will be prompted to save the file if you have made any changes to it. You are always warned if you are potentially going to lose a layout or schematic.

Defining the working directories

BOARDMAKER gives you a great deal of flexibility in how you set up the working directories for your files. Using the {Config}{Directory setup} option you can define the DOS path for the current, PCB, schematic, library and net file directories. These can be organized so that BOARDMAKER is project based, but you must fully understand the nature of DOS directories.

For more details of how to set these directories see DIRECTORY SETUP in the Command Reference.

The Backup timer

BOARDMAKER provides an alarm backup timer which can be set to remind you to save your work. This alarm will go off after a time which you set in {Config}{Backup timer}. This can be set between off and 30 minutes. You will then be prompted to save the file, exactly as if you had selected {File}{Save}.

Using and organizing symbol libraries

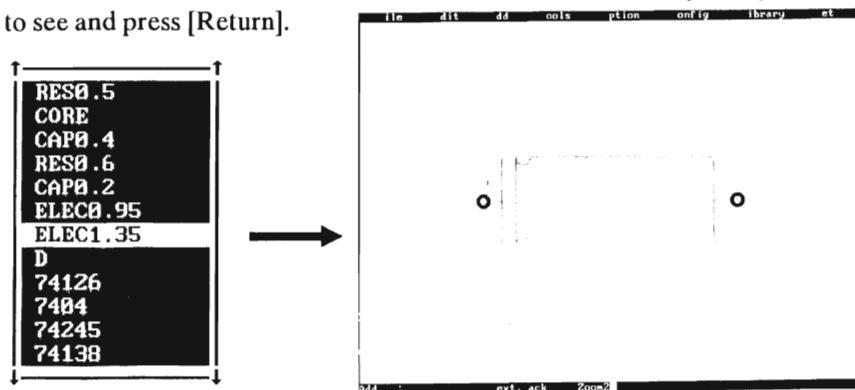
Symbol libraries are used to store a collection of user defined PCB layout or schematic shapes which can be recalled while you are using the layout or schematic editor. To use symbol libraries in BOARDMAKER, you will need to understand the basic principles of how to create new symbols and libraries as well as how to recall the symbols when you need them.

Symbol library editor

Symbols can only be created using the symbol library editor, accessible from the main menu. The library editor is identical in use to the layout or schematic editor, and you use it by placing any combination of tracks, pads or text as normal, but instead of saving the file, you add it to a library. Although you can save approximately 250 equivalent ICs in any library.

Browsing libraries

Before we go on to creating or editing symbols, we need to know what libraries are in the library directory on your disc and what symbols are in each. Select {Library}{Browse libraries} from either the layout, schematic or library editor and the {Option} menu will present you with the names of the symbols in the currently selected library; at the bottom of the list, the names of any other libraries will be shown. Move the banner over the name of the symbol you want to see and press [Return].



The screen will clear - you will not lose your work - and the symbol will be presented with a small white cross indicating its origin. The name of the symbol will be displayed in the status banner along with the two options BACK and NEXT. Pressing [B] will display the previous symbol and [N] will display the next symbol. By pressing [Esc] you can get back to the option menu first displayed. To examine a different library, move the banner over its name in the option menu and press [Return] and the contents of that library will then be displayed.

Viewing a library

This provides a method of viewing the contents of a named symbol library in textual form. Select {Library}{View library} and you will be prompted :

>Enter library file name :

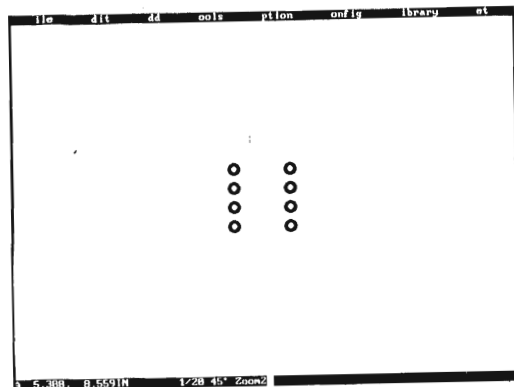
The default library name will be displayed as the last library name used. If no name has been specified then the default name SYMBOL.LL or SYMBOL.SL will appear - these are the default names for the layout and schematic library respectively. Enter the name you want and press [Return]. You will then see a list of entries in that library. If you want to print this list, simply press [Print Screen] and it will be sent to the printer selected by {Config}{Printer settings}{Output to ..}.

Creating a new symbol

When you select {Library editor} from the main menu, you will need to choose whether you want to create or edit a PCB layout or schematic symbol - the two are saved in different libraries. Creating a new symbol basically involves placing the required pads and silk screen details, including a designator and then naming each of the pads or pins - this is then saved into a specified library.

We will use the example of designing a PCB layout symbol for an LM741 op-amp as an example. Select {Library editor}{PCB library} from the main menu to get into the library editor and choose zoom level 2 by pressing [2]. This component has eight pins so place eight pads as shown below. If you are unsure about how to add pads see the section entitled Placing pads.

You now need to add a silk screen on layer 0, the top silk screen layer. Use {Add}{Track} to place a track on layer 0 as shown.



This has established the basic pattern for the symbol - we must now add a designator prefix. This is the name which will appear on either of the silk screen layers - in this case 'IC' will be used. When the component is subsequently loaded in the PCB layout editor you can modify this in terms of textual content, size and orientation.

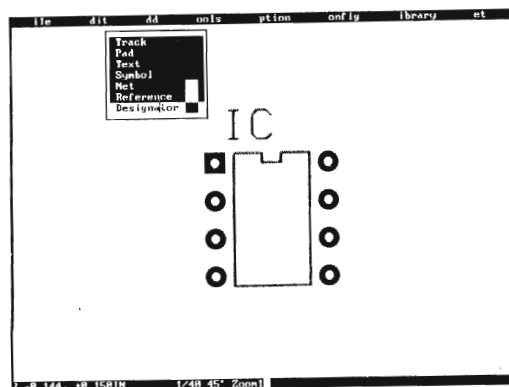
Also, BOARDMAKER will automatically append and increment a number to it as you place it.

Select {Add}{Designator} and you are prompted:

>Position cursor and press [Return]

Put the cursor where you want the designator to be, press [Return] and you are prompted for the designator name:

>Designator prefix :



Each designator prefix can be any alpha-numeric string on the top or bottom silk screen and would typically be text such as IC, R or TR. You add the actual component number when you are in the PCB editor - you also have control over its size, width etc.

Enter 'IC' [Return] to place it. If you are not happy with

the position of the text, it is still mobile and can be moved by simply moving the cursor and pressing [Return]. When you have finished, press [Esc] to fix it in position.

To distinguish between ordinary text in a symbol and designator text in the library editor, the designator will be displayed in a box which will not appear on your layout.

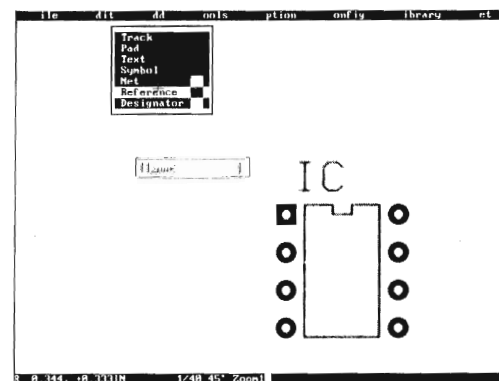
The final task is to give each of the pads or pins a unique name in order that we can explicitly identify them in relation to nets. This is done by adding a reference point to each pad. Select {Add}{Reference} and you are prompted

>Position cursor and press [Return]

Put the cursor on pin 1 and press [Return]. You are now prompted :

>Reference name :

You are being asked to enter a unique name for this pin. Enter '1' [Return] and a box will appear which is connected to pin 1 as shown below.

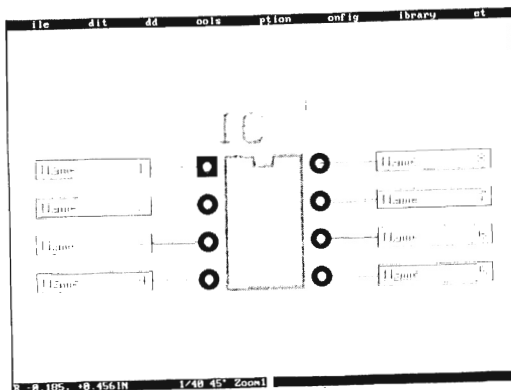


If you don't like the position of the box, then simply put the cursor where you want it and press [B] or select {Option}{Reference box}. If you want to reference it to another pin, then put the cursor on that pin and press [Return].

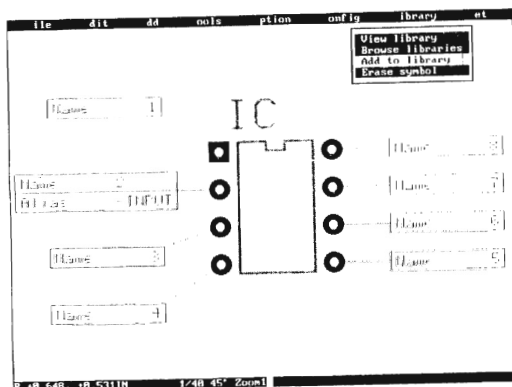
This indicates that pin 1 is referenced by the name '1'.

The name you give to the reference does not have to be a number. For example, if you were designing a transistor symbol you might call one of the pads BASE or EMITTER. The reference name is the only mandatory information required for symbol pins, although you can add further information as detailed later.

To create references for the remaining seven pins, you could follow the above procedure, but as usual with BOARDMAKER there are built in facilities to ease the process. Put the cursor on pin 2 and press [R] to repeat the reference or select {Option}{Repeat} - a duplicate will appear connected to pin 2. Press [I] or select {Option}{Increment} and the name will increment to 2. You can repeat this for each of the remaining pins until it appears as like this :



Since this is the inverting input we can label it as such. Enter 'INPUT' [Return] and the pin will be given an alias. You may want to repeat this for the non-inverting input (pin 3) and the supply pins (4 and 7).



These are the basics of designators and references. More information can be added to each pin such as the connect mode and the force layer. For full details, see the command reference under ADD DESIGNATOR and ADD REFERENCE.

Now you have finished you will need to set the origin for the symbol. This is the point which will be at the cursor position when you are placing the symbol in a drawing, so make the origin a sensible point e.g. pin 1 on a dual-in-line package. Use the SET ORIGIN [O] command by selecting {Tools}{Set origin}. Now select {Library}{Add to library}. You will then be prompted :

>Have you set the origin ? (Y/N)

If you have then enter [Y], if not or you want to change it, then enter [N]. You will then be prompted for the name you want to give to the symbol :

>Enter symbol name :

Although you have given the pins unique names you can also add an alias which might be the function of the pin. Select {Edit}{Reference} and when prompted, put the cursor over the reference box associated with pin 2. Choose {Option}{Alias} and you are prompted :

>Editing alias :

Enter a NEW name followed by [Return]. You will then be prompted to enter the name of the library you want to save the symbol in.

>Enter library filename :

Enter the name of an existing library followed by [Return].

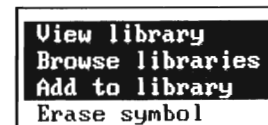
Creating a new library

If you want to create a new library, then simply delete name presented in the line editor and enter a new name. You will then be asked to confirm you want to create the new library.

>Create new library? (Y/N)

Enter [Y] to create the new library.

Erasing a symbol



At some time you may need to delete a symbol from a library. Select {Library}{Erase symbol} and you will be prompted for the name of the symbol you want to delete :

>Enter symbol name :

Enter the name followed by [Return]. You then need to enter the library you want to erase the symbol from :

>Enter library filename :

You will then be asked to confirm the deletion of the symbol :

>Delete library entry ? (Y/N)

Enter [Y] to delete the symbol or [N] if you change your mind.

Remember, you can only do this while you are in the library editor.

Changing an existing symbol

Whilst placing a symbol in the layout or schematic editor you may decide that you want to edit the symbol and change it in some way. Since this can only be done in the library editor, you will need to save your drawing and exit to the main menu and select the relevant library editor. Now use {Add}{Symbol} to place the symbol you want to alter. Although this seems identical to placing a symbol in the layout or schematic editor, you will find that the symbol has been 'exploded' into its component parts which you can edit as required. When you have finished making the changes you will need to add it back into the library with the same name. Select {Library}{Add to library} and follow the prompts to save the symbol as before. This time however, you will be prompted :

>Overwrite existing symbol ? (Y/N)

View library
Browse libraries
Add to library
Erase symbol
Update symbols
Export symbol

You will now need to go back to your drawing to update the symbols you have already placed. Exit the library editor, select the editor you used earlier and load the drawing you want to update the symbols on. Select {Library}{Update symbols} and the {Option} menu will present a list of all the symbols in the drawing. Select the one you want to update and press [Return].

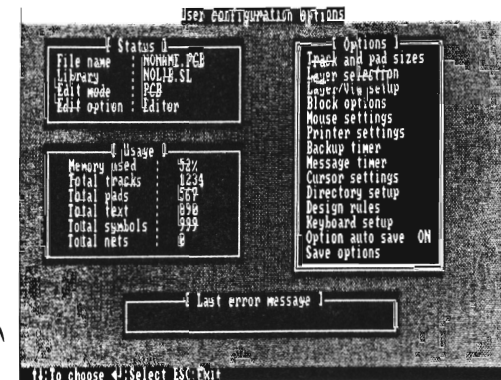
You are then prompted for the library you want to update from.

>Enter library filename :

Type in the name of the library the symbol is in and press [Return]. The message 'Symbols have been updated from library' will appear indicating that ALL occurrences of the symbol in this drawing have been updated with the new version you created in the library editor.

Changing the configuration

The {Config} screen contains options which allow you to change the configuration of BOARDMAKER. It also allows you to save the settings so that the next time you use it, the setup will be the same. For details of exactly



what is saved see SAVE OPTIONS in the Command reference, but generally, any items you can change in the {Config} screen will be saved in a file called CONFIG.BM in the current directory.

This section explains the options which have not already been covered previously.

X and Y mouse locking

You can force the cursor to move only in the X or Y direction when using the mouse, allowing you to easily align items on a drawing. By pressing the [X] key you lock the X position and the cursor will only move in the Y direction. Similarly, the [Y] key will lock the cursor Y direction. NB these keys must be held down whilst the mouse is being moved.

Mouse settings

This option allows you to set the mouse pointer sensitivity between very fast and very slow, as well as the type of mouse you are using - 2 button or 3 button. If you are using a 3 button mouse, the centre button acts as a PAN key.

Message timer

BOARDMAKER displays messages on the status banner for a user definable time. This option allows you to set the time - if you are an experienced user you can turn them off altogether.

Config auto save

When set to ON, this option will automatically save the configuration when you exit BOARDMAKER. You can explicitly save by selecting {Save options}.

Keyboard settings

By selecting {Config}{Keyboard settings}{Keyboard layout} you can change the way in which the function keys are implemented to take advantage of an XT or AT keyboard layout.

The {Country} option allows you to select the relevant keyboard driver for particular countries. To select a driver, click on this option until the required country name appears.

Output facilities

Any PCB layout software is only as good as its output facilities - crisp, accurate outputs are the ingredients required for professional board production. Integrated into the BOARDMAKER package are facilities which allow you to produce draft and final quality artwork using a dot-matrix printer, laser printer or penplotter. You can also generate the necessary Gerber format photoplot files to send to a photoplot bureau and Excellon NC drill files which are sent with your final artwork to a PCB manufacturer. These choices allow you to select both the quality and cost to produce your PCB.

You will find that there are some differences in the options available, depending on whether you are outputting a schematic or layout. Some options, for example, solder resist mask enlargement, are not applicable for schematics.

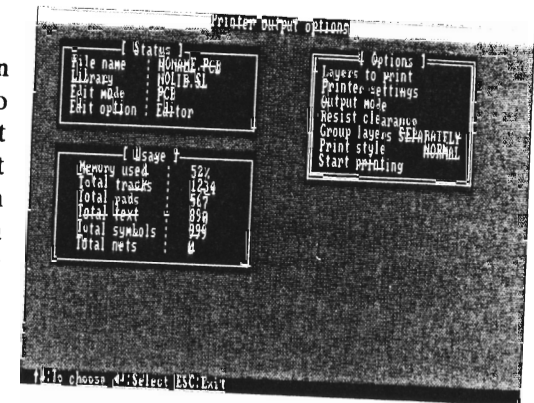
Dot-matrix printer

When you select {Output}{Print all} from the layout or schematic editor, you will be presented with the Printer output options screen. This allows you to setup BOARDMAKER prior to printing. You will also get here when you are printing a block - the setup options are identical in each case. Dot-matrix output

can be used to generate camera ready artwork by printing at twice or four times actual size and photo reducing the result.

Layers to print

The {Layers to print} option gives you the opportunity to select which layers you want to be printed. If you select more than one layer, they can be printed in sequence on separate sheets of paper or on top of each other for checking purposes.



Printer settings

You can use the {Printer settings} option to setup :

- ☐ the {Printer type}. A 9 pin, 24 pin or Laser printer can be used. Laser gives the best results and a 24 pin printer will produce output about ten times faster than a 9 pin printer.
- ☐ the {Printer width}. This selects the carriage width for your printer. You will be warned if the drawing cannot be made to fit.
- ☐ the {Printout scale}. This can be set to 1:1, 2:1 or 4:1 for a 9 pin printer, 1:1 or 2:1 for a 24 pin printer or 1:1 for a laser printer. BOARDMAKER will not allow you set and invalid combination of output scale and printer type.
- ☐ the {Print mode}. Use the {Normal} setting for fast draft prints or the {Bold} setting for final artwork as required.
- ☐ the Output port. You can choose between the parallel port LPT1, the serial port COM1 or FILE to send it to a named file.

Output mode

This determines the type of output produced. {Normal} will print all the layers, including tracks, pads, text and symbols. {Drilling template} will print the pads only, to be used as a drilling guide. {Solder resist mask} will produce the pad areas suitably enlarged, with their holes filled in.

Resist clearance

This allows you determine the amount of clearance you want the solder resist mask to have for the {Output mode} setting above.

Group layers

This determines whether the layers you are going to output are grouped together i.e. on top of each other or separately on different pages. This is only available just prior to printing.

Output style

This option allows you to invert the printer output and directly create negative artwork. The {Print style} option toggles this to be either NORMAL or INVERTED.

Laser printers

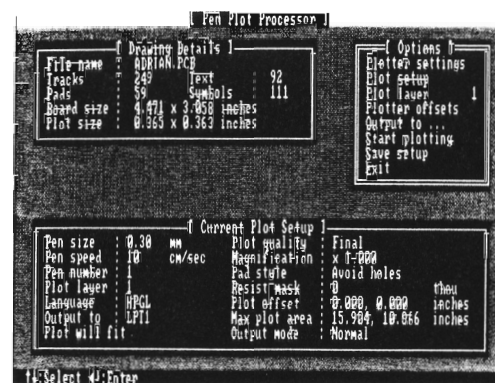
The settings for the laser printer output is similar to the above dot-matrix selections. Except for :

Printer settings

The {Laser scaling} option allows you to independently scale the X and Y axes to account for any inaccuracies in your laser printer.

Penplotter

BOARDMAKER is capable of producing quality artwork on a penplotter using HPGL, or DMPL plotting languages. When you select



{Output} {Penplot All} from the layout or schematic editor you will be presented with the Pen Plotter options screen, which allows you to select all the plot and plotter settings. If you selected {Penplot block} in block mode from the editor, you will also be presented with this options screen.

Drawing details

The drawing details window shows the name of the file you are working on, along with the number of tracks, pads, text and symbols. Also shown are the size of the board and the size of the plot. BOARDMAKER will automatically calculate the size of the board for you, regardless of its actual position in the drawing area. The plot size represents the size the finished plot will be, taking into account the magnification factor etc

Current plot setup

This window shows the settings currently made. These will change dynamically as you change them from the option menu. It also tells you whether the plot you are about to make will fit or not.

Plotter settings

The {Plotter settings} option allows you to configure BOARDMAKER to suit your plotter. You can change the :

- ☐ pen size
- ☐ pen speed
- ☐ pen number
- ☐ plotter language

- ☐ Maximum plot area. You use this option to enter the lower left and upper right co-ordinates of your plotter. These have to be entered in HPGL units which you will find in your plotter manual.

Plot setup

The {Plot setup} option allows you to determine the type of plot you want to produce. You can change the :

- ☐ plot quality. Select either {Draft} for a quick check plot or {Final} for the final quality artwork.
- ☐ pad hole type. You have the choice of whether to fill or avoid the holes in the pads.
- ☐ plot mode. This determines the type of output produced. {Normal} will print all the layers, including tracks, pads, text and symbols. {Drilling template} will print the pads only, to be used as a drilling guide. {Solder resist mask} will produce the pad areas suitably enlarged, with their holes filled in.
- ☐ magnification factor. Use this to change the scale of plot you require. This can be set in the range 0.1 to 10.0 - enter 1.0 for a 1:1 plot.
- ☐ solder resist mask. This allows you determine the amount of clearance you want the solder resist mask to have for the {Plot mode} setting above.

Choosing the layer to plot

As you select the {Plot layer} option, you increment the layer that will be plotted, both in the menu itself and the Current plot setup window - click on it until the layer you want to plot is displayed.

Output to ...

The {Output to..} option lets you determine where the output is sent to. This can be one of the parallel or serial ports, whichever your plotter is connected to, or you can send the output to a file.

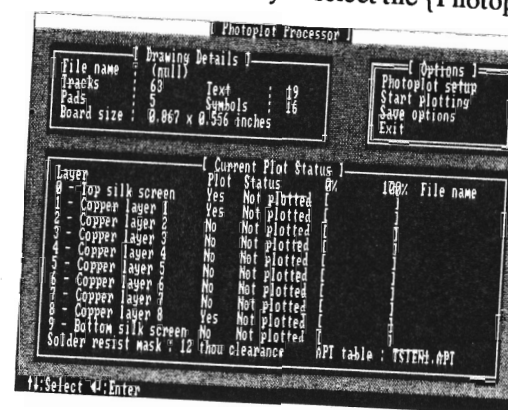
You can also set up the serial port communication protocol by choosing {COM1/COM2 setup}. This allows you to set up the bit rate, number of data bits, number of stop bits and parity.

Save setup

This option lets you save the settings you have made.

Photoplotter

Using the {Photoplot} output option allows you to generate a Gerber format data file which you can then send to a photoplot bureau via post or modem. This produces very accurate results and should be considered for your final quality output. When you select the {Photoplot} option from the layout editor,



you will be presented with the photoplot setup screen. Use the options on this screen to specify your output requirements. You can opt to use either FIXED or VARIABLE aperture tables to suit your plotting. With VARIABLE apertures, BOARDMAKER assigns the apertures which are used and generates an aperture table detailing the ones which have been used. With FIXED apertures, you provide an aperture table which tell BOARDMAKER what apertures are available. For full details of how to use this facility see PHOTOPLOT in the command reference.

Drawing details

The drawing details window shows the name of the file you are working on, along with the number of tracks, pads, text and symbols.

Plot status

This window shows the settings you have currently made along with the current status of the plot.

Solder resist mask

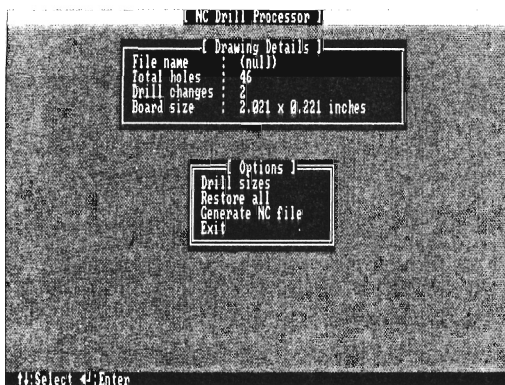
Use the {Solder resist mask} option to choose the solder resist clearance you require.

Layer to plot

Use the {Layers to plot} option to select which layers you want to plot. You will not be able to turn a layer on which has nothing on it.

NC drill

Using the {NC drill} output option allows you to generate Excellon ASCII format drilling data which you can then send to your PCB manufacturer along with your final artwork. When you select the {NC drill} option from the layout editor, you will be presented with the NC setup screen. Use the options on this screen to specify your output requirements.



Drawing details

The drawing details window shows the name of the file you are working on, along with the number of holes on the board and the number of drill changes required. The size of the board is also displayed. You will find these details useful when you are getting quotes from PCB manufacturers.

Drill sizes

When you select {Drill sizes} from the menu, you will be presented with a table of all the available drill sizes BOARDMAKER has to offer, along with the number of holes of each size. Sizes that have been used are displayed highlighted.

Changing the hole sizes

If you move the cursor to any of the used hole sizes and press [Return], an edit window will appear which allows you to change the hole size - the original size is also given for reference. You can use this option to reduce the number of drill changes.

Restore hole sizes

If you have changed hole sizes, you can restore them all back to their original value by selecting {Restore all}.

Chapter 4

Command Reference

This chapter is a complete reference to all the BOARDMAKER commands. Each command is presented separately and in alphabetical order, except the first entry, which is a guide to using the command reference.

THE COMMAND REFERENCE

Purpose	This gives a brief summary of what the command does.
Mode	<p>The mode for the command described. These modes are</p> <ul style="list-style-type: none">Track - relevant to tracks or linesPad - relevant to padsText - relevant to textSymbol - relevant to symbolsHilite - relevant to net highlightingNet - relevant to net editingDesig - relevant to designator in PCB library editorRef - relevant to references in PCB library editorAll - available at all times
Description	This describes what the command does, how it works and gives any details you need to use the command and any related functions.
Keys	<p>This tells you the keys to press to directly access the command from the keyboard if available. It is possible to configure BOARDMAKER to take advantage to either an XT or AT function key layout, so where appropriate, this heading gives both keys in the form :</p> <p>[F2]-(XT) or [F5]-(AT)</p> <p>meaning, function key F2 for the XT setup or function key F5 for the AT setup.</p>
Menu	<p>This gives you the menu selections to make to access the command described from the menu banner at the top of the screen. They are shown in the following format :</p> <p>{ Config } { Keyboard settings }</p>

meaning, put the cursor on the menu banner item Config and press [Return] or click the left hand mouse button, then select the Keyboard settings option from the menu presented.

See also List other related commands you may wish to read about.

Example Gives a concise example of the command usage.

ADD DESIGNATOR

Purpose To add a new designator prefix to a symbol

Mode Desig

Requirements You must be in the PCB library editor

Description This command provides an method of adding a designator prefix to a symbol when you are in the library editor. For example, if you had just designed a symbol for a 14 pin dual-in-line package, this command allows you to determine the default text and position for the silk screen legend starting with the letters 'IC'. If you use this facility, it means you can take full advantage of BOARDMAKER's ability to auto-increment designator numbers. Also, when you move the symbol to a new position in the PCB editor, the designator will stay with the symbol.

Each designator prefix can be any alpha-numeric string on the top or bottom silk screen and would typically be text such as IC, R or TR. You add the actual component number when you are in the PCB editor - you also have control over its size, width etc.

To distinguish between ordinary text in a symbol and designator text in the library editor, the designator will be displayed in a box which will not appear on your layout.

To place a designator prefix, position the cursor where you want it, select ADD DESIGNATOR and you will be prompted :

> Designator prefix :

Enter the text you want followed by [Return] to place it. If you are not happy with the position of the text, it is still mobile and can be moved by simply moving the cursor and pressing [Return]. When you have finished, press [Esc] to fix it in position.

Keys [Shift][F6]-(XT) or [Shift][F7]-(AT)

or

[Ctrl][A]{Designator}

Menu {Add}{Designator}

Warnings > Only one designator allowed.

You can only have one designator prefix per symbol. Also, symbols which have references **must** have a designator prefix.

See also EDIT REFERENCE, ANTICLOCKWISE, CURRENT, KILL, LAYER, MIRROR, SIZE, TEXT CONTENT, WIDTH

ADD NET

Purpose To add a new net

Mode All

Description This command is part of the graphical net editor. It allows you to specify which component pins or nodes are to be connected together when you come to route the layout. This set of connections or net allows you to take full advantage of BOARDMAKER's ability to quickly and accurately route the board, without continual reference your circuit schematic - you will also be able to use the FULL DESIGN RULE CHECKER to automatically check the entire layout against the connections specified.

To use this facility, it is recommended that you first load the symbols you require into the PCB editor and give them designators. The actual position of the symbols is not important for now since you are only specifying the connections at this stage.

To create or add a new net, select {Add}{Net} and move the cursor over the point you want on the net and press [Return]. You are then prompted for the name of the net.

> Editing net name :

Enter a unique alpha-numeric string of up to 16 characters. This is the name by which this collection of nodes will be known so it is best to choose something which has some relevance e.g. VCC, ADDR0, +12V. If there is no sensible name for the net then use the default net name which BOARDMAKER will automatically supply and increment for you - this will be in the form N00001.

To build up the net, you can simply move the cursor to each point you want to be on the net and press [Return]. Once again, BOARDMAKER can ease you through this process by presenting you with details of the components and their

pins. Once you have started a new net, select {Option}{Add to by name} and you are prompted :

> Enter Designator, Pin name :

You can now enter the next point you want to be on the net in the form IC10,4 [Return] and this node will be added to the net - the screen will pan to the point if necessary. If simply press [Return] in reply to the prompt, the {Option} menu will present a list of all the component designators available. Put the banner over the one you require, press [Return] and you then get a list of all the pin names on that component.

This procedure is designed to be intuitive so that you can simply enter a designator name at the prompt and the {Option} menu will show you a list of the pins on the it.

Small diagonal white crosses will appear at the net nodes and they will be joined together by white lines which indicate the nets. When you have finished, press [Esc] or the right hand mouse button. You can then move to another point to start another net. Continue this procedure to quickly build up a netlist for the entire layout.

You do not need to worry about the order you select the nodes, this can be changed and arrangement of the ratsnest tidied up using OPTIMIZE NETS.

Warnings

> This node is already part of a net

Nodes can only be part of one net, either choose another point or delete the offending node from your netlist.

> Net name must be unique

The net name you typed in already exists, choose another name.

> Must not connect this node

When the symbol was designed, this node was defined as NEVER CONNECT

Keys

[Shift][F2] - (XT) [Shift][F6] - (AT)

Menu

{Add}{Net}

See also

EDIT NET, CURRENT, HOME, END, NEXT, BACK, NET NAME, KILL, OPTIMIZE

Example

- ☐ In the PCB editor select {Add}{Symbol} and enter 'DIP14' from the standard library 'SYMBOL.LL'
- ☐ Select {Add}{Net} and put the cursor on pin 1 of the symbol and press [Return]. You are then prompted for the net name - enter the name '+12V'[Return].
- ☐ Move the cursor over pin 2 and again press [Return]. A thin white cross will be drawn on pins 1 and 2 indicating they are the nodes on this net - a thin white line will be drawn between them to indicate the connection.
- ☐ Repeat this procedure to add more pins to the net '+12V'
- ☐ When you have finished press [Esc]

ADD PAD

Purpose	To add new pads.
Mode	All
Description	This command provides a method of adding new pads to a drawing. To lay a pad, position the cursor where you want the pad, select ADD PAD and press [Return] to place it. If you are not happy with the position of the pad, it is still mobile and can be moved by simply moving the cursor and pressing [Return]. When you have finished, press [Esc] to fix it in position.
Keys	[F4]-(XT) or [F6]-(AT) [Ctrl][A]{Pad}
Menu	{Add}{Pad}
See also	ANTICLOCKWISE, CURRENT, KILL, LAYER, AUTOVIA, MIRROR, REPEAT, SHAPE, WIDTH

ADD REFERENCE

Purpose	To add new reference to a symbol
Mode	All
Requirements	You must be in the PCB library editor
Description	<p>This command provides an method of adding a reference to a symbol when you are in the library editor. A reference is a label which gives information about a pin on a component or symbol which about it. For example, when designing a symbol for a transistor, it will probably consist of three pads and a silk screen legend. To this, you can add the default position of the component designator using ADD Designator prefix. To take full advantage of BOARDMAKER's ability to automatically check the PCB, you need some method of labelling the transistor pads.</p> <p>The position of the boxes is un-important, it is the association of the reference to the particular pad or pin which explicitly tags the information.</p> <p>These reference boxes are only displayed while in the library editor to give an immediate picture of the names you have given to the pads of a symbol. When you select {Add}{Reference} you are prompted :</p> <p>> Reference name :</p> <p>The name you enter can be any alpha-numeric string up to 16 characters - this can either be the number of the pin or the name you want to associate with the pin. If you plan to use references, the reference name is mandatory and must be unique as far as the symbol is concerned. When the name is entered, you will see the reference box appear which contains the name you just entered. This must now be associated with a particular pad, so move the cursor over the pad you are naming and press [Return] - a line will now be drawn between the reference box and the selected pad.</p>

You now have the option to add further information to the reference or move the reference box to a new position using **REFERENCE BOX**. Other options available function as follows :

ALIAS - To add a secondary name to the pin e.g. Pin name = 14, Pin alias = Vcc

CONNECT MODE - To indicate whether the pin should ALWAYS be connected, NEVER connected or you DON'T CARE whether it is connected. This information is used by the design rule checker to warn you if you incorrectly connect a track to a pin.

FORCE LAYER - Normally, pads are on all layers, but symbols for surface mount devices and edge connectors have the pads associated only with one layer - the force layer. This information eliminates any confusion when you use the design rule checker.

The quickest way to add a large number of references, say for a 28 pin IC, is to create one initial reference, with a pin name and alias and then copy it using **REPEAT** and change the name and alias using the **INCREMENT** and **DECREMENT** functions.

If the display becomes cluttered, you can use the **SIZE** function to switch between two reference box sizes.

Keys [Shift][F4]

Menu {Add}{Reference}

See also ADD DESIGNATOR, EDIT REFERENCE, NAME, ALIAS, REFERENCE BOX, FORCE LAYER, CONNECT MODE, INCREMENT ALIAS, INCREMENT NAME, DECREMENT, ALIAS, DECREMENT NAME, SIZE

ADD SYMBOL

Purpose To add a new symbol.

Mode All

Description This command provides a method of adding new symbols to a drawing. To place a symbol, position the cursor, select **ADD SYMBOL** and press [Return] to place it. If you are not happy with the position of the symbol, it is still mobile and can be moved by simply moving the cursor and pressing [Return]. When you have finished, press [Esc] to fix it in position.

If you cannot remember the names of the symbols in your libraries, simply press [Return] when you are prompted for the symbol and their names will be presented in the {Option} menu.

If the symbol you are adding was given a designator prefix when designed, this will automatically increment. e.g. if the prefix was 'R' it will increment to 'R1'.

Keys [F8]

[Ctrl][A]{Symbol}

Menu {Add}{Symbol}

See also ANTICLOCKWISE, CURRENT, KILL, MIRROR, REPEAT

ADD TEXT

Purpose	To add new text.
Mode	All
Description	This command provides an method of add new text to a drawing. To write text, position the cursor where you want it, select ADD TEXT and you are prompted : <div style="margin-left: 20px;">> Editing text :</div> Enter the text you want followed by [Return] to place it. If you are not happy with the position of the text, it is still mobile and can be moved by simply moving the cursor and pressing [Return]. When you have finished, press [Esc] to fix it in position.
Keys	[F6]-(XT) or [F7]-(AT) [Ctrl][A]{Text}
Menu	{Add}{Text}
See also	ANTICLOCKWISE, CURRENT, KILL, LAYER, MIRROR, REPEAT, SIZE, TEXT CONTENT, WIDTH

ADD TRACK

Purpose	To add new tracks or lines.
Mode	All
Description	This command provides a method of adding new tracks or lines to a drawing. To lay a track, position the cursor at the starting point of a track segment, select ADD TRACK, move the cursor and press [Return] to place the segment. To add more track segments, simply move the cursor again and press [Return]. This is repeated until you have placed all the segments you want and [Esc] is pressed to finish.
Keys	[F2]-(XT) or [F5]-(AT) [Ctrl][A]{Track}
Menu	{Add}{Track}
See also	BACK, CURRENT, DELETE, END, ASSIST FLIP, HOME, INVERT ORDER, KILL, SEGMENT LAYER, AUTOVIA, ALL LAYERS, INSERT/MOVE NODE, NEXT, REPEAT, SPLIT, SHIFT, ALL WIDTH, SEGMENT WIDTH

ADD TO LIBRARY

Purpose	Adds a new symbol to a library.
Mode	Symbol library
Description	<p>When a new symbol has been created in the symbol library editor, this command allows you to add the symbol to a library. When selected, you will be asked for confirmation that the origin for the symbol has been set. See SET ORIGIN for more details. The origin should be chosen at some sensible point on the symbol e.g. pin 1 on an IC. This point will be under the cursor when the symbol is later loaded into a layout or schematic. You will now be prompted for the name of the symbol and the library that you wish to save it in.</p> <p>To create a new library, enter its name when prompted and confirm that you are creating a new library. This procedure allows you to have many libraries, allowing you to perhaps have project libraries associated with a number of PCBs and schematics.</p> <p>If the symbol name already exists in the library, for example when modifying an existing symbol, you will be prompted 'Overwrite existing symbol? Y/N'.</p> <p>BOARDMAKER allows you to have approximately 250 symbols in any one library. The exact maximum number cannot be defined since the size of the symbols may vary and the library size is currently restricted to 64K.</p>
Requirements	Must be in the symbol library editor.
Keys	[Ctrl][L]{Add to library}
Menu	{File}{Symbol library...}{Add to library} for all displays {Library}{Add to library} for EGA and VGA displays
See also	BROWSE LIBRARIES, VIEW LIBRARY, ERASE SYMBOL, UPDATE SYMBOLS, SET ORIGIN

Example

- ☐ Select the library editor (Layout mode) from the main opening menu.
- ☐ Select Add Symbol by pressing [F8].
- ☐ At the symbol name prompt type in 'DIP14'.
- ☐ At the symbol filename prompt type 'SYMBOL.LL' - this is the standard issue library package.
- ☐ The display will now show the DIP14 component which for the purpose of this example we will save under another name.
- ☐ Put the cursor on pin 1 of the DIP14 and press [O] to set the origin.
- ☐ Press [Control][L] to access the library menu and select Add to library. You will be prompted to set the origin, type 'Y' since you have just done it and enter the symbol name as 'TEMP' [Return]. This can be deleted later if required.
- ☐ At the symbol filename prompt press [Return] because BOARDMAKER defaults to the last used library name.
- ☐ The symbol will now have been added to the library under the name TEMP and may always be referred to by this name.

ALIAS

Purpose	To add or edit a references alias
Mode	Ref
Requirements	You must be in the PCB library editor
Description	<p>When designing symbols which use references the only mandatory information is the reference name e.g BASE, EMITTER etc. However, it is sometimes beneficial to give a pin a secondary name. When designing symbols for ICs, it is usual to make the reference name the pin number and the pin alias the function for that pin. For example, Pin name = 5, Pin alias = CLOCK. This extra information means you can easily see where any errors have occurred in the layout when you use the design rule checker.</p> <p>To add or edit the alias, you must first select the reference box by editing it {Edit}{Reference}. When you select {Option}{Alias} you will be prompted :</p> <p>> Editing alias :</p> <p>You can enter any alpha-numeric string up to 16 characters long. Press [Return] when you have entered the alias and it will appear in the reference box which will autoscale to accommodate the text.</p> <p>For a full description of references, see ADD REFERENCE</p>
Keys	[A]
Menu	{Option}{Alias}
See also	EDIT REFERENCE, NAME, FORCE LAYER, CONNECT MODE, INCREMENT ALIAS, DECREMENT ALIAS

ALL LAYERS

Purpose	To move all the track segments of the current track to a new layer.
Mode	Track
Description	<p>When editing an existing track or while adding a new track on anything but a single layer board, you will at some time need to change the layer that the track is on. BOARDMAKER provides a neat and efficient way of switching layers which allows you to change either the layer for the current and subsequent track segments or the layer for the whole track.</p> <p>When you are editing or adding a track you may need to switch the whole track to a different layer in which case you use the ALL LAYERS command. You will be prompted for the new layer you want to put the track on.</p> <p>BOARDMAKER provides a total of ten usable layers of which eight are specifically designated as copper or track layers - the other two are nominally used for the top and bottom silk screen. Although eight copper layers are provided, the majority of PCBs you are likely to design will not use all the layers for the final board. Consequently, you can make good use of the spare layers for temporary work space, mechanical details, drilling idents or any other information relevant to the layout.</p> <p>The layers are nominally designated as follows :</p> <ul style="list-style-type: none"> Layer 0 - Upper silk screen Layer 1 - Component side Layer 2 - Copper layer 2 Layer 3 - Copper layer 3 Layer 4 - Copper layer 4 Layer 5 - Copper layer 5 Layer 6 - Copper layer 6 Layer 7 - Copper layer 7

Layer 8 - Non-component side

Layer 9 - Lower silk screen

You choose which layers are displayed using the {Layer selection} option in the {Config} menu. Any layer can be given any of the available colours, although the number of colours available depends on the type of display adaptor you are using. CGA display users only have a choice of three colours whereas EGA and VGA users have a choice of six colours. The colours available have been chosen so that when lines or tracks cross on the screen you will see a different colour at the point where they cross. This ensures that one track cannot be easily hidden under another. Note that any item on the board which is on all layers will always be shown in white. Also, if you try to place an item on a layer that is not turned on, you will be warned and the item will not be moved to that layer. In this case you will need to turn the layer on using the {Layer selection} option in the {Config} menu.

- Keys**
- [Shift][L]
- Menu**
- {Option}{All layers}
- See also**
- SEGMENT LAYER, LAYER SELECTION, AUTOVIA
- Warnings**
- > Selected layer is not enabled
- Example**
- You have tried to put a track on a layer which is not enabled.

☐ Select Add track by pressing [F2]-(XT) or [F5]-(AT) and note the information banner shows the layer of the track.

☐ Move the cursor and press [Return] to place a track segment and note the colour of the track.

☐ Repeat this a few times to create a number of track segments.

☐ Select ALL LAYERS by pressing [Shift][L] and you will be prompted for the new layer you want to use, so enter the new layer number (0-9).

☐ The display will now show the whole track in the colour of the new layer.

ALL WIDTH

- Purpose**
- To change the width of a complete track.
- Mode**
- Track
- Description**
- The command allows you to change the width of every track segment in the current track so providing a method of quickly increasing or decreasing the width of an existing track or of the track you are currently creating. When selected, you will be prompted for the track width number in the range 1-8 and the information banner will display the current track width in the form Width = 1(2) indicating track width number 1, which is set to 2 thou width. To see what the actual track widths are in relation to the track width number you will need to look at the TRACK AND PAD SIZES display in the {Config} menu.

If you need to change the width of an individual track segment rather than the whole track, use the SEGMENT WIDTH command.

- Keys**
- [W]
- Menu**
- {Option}{All width}
- See also**
- SEGMENT WIDTH, TRACK AND PAD SIZES, WIDTH, ALL LAYER, SEGMENT LAYER
- Example**
- ☐ Select Add track by pressing [F2]-(XT) or [F5]-(AT) and place a few track segments by moving the cursor and pressing [Return]. Note the track size displayed on the information banner in the form Width = 1(2) indicating track width number 1 which has a width set to 2 thou.

☐ Now change the whole of the track size by selecting ALL WIDTH [W] and enter '8' [Return] as the track width number.

☐ The track will now have changed to the width set for track width number 8 and the actual size will be displayed on the information banner.

ANGLE ASSIST

Purpose	To assist in the placement of tracks at 45 and 90 degrees.
Mode	Track
Description	When placing tracks on a printed circuit board it is usual to use either 45 or 90 degree corners when changing direction. This command, which can of course be turned off if not required, allows the user to easily determine where the next track segment would be placed before actually fixing it into position. The status banner displays the current angle assist setting which can be either 45, 90 or off and is changed by pressing [Shift][A] until the required angle is shown. The point at which the corner is positioned can be changed to one of two positions using ASSIST FLIP.
See also	ADD TRACK, EDIT TRACK, ASSIST FLIP, RUBBERBAND, QUASI-TRACK
Keys	[Shift][A]
Menu	{Option}{Angle assist} <input type="checkbox"/> Select Add Track by pressing [F2]-(XT) or [F5]-(AT). <input type="checkbox"/> Press [Shift][A] until the status banner displays 45 degrees. Make sure rubberbanding is enabled using [Shift][B]. <input type="checkbox"/> Move the cursor around and notice that two track segments are displayed on the screen with a 45 degree angle between them. <input type="checkbox"/> Press [Return] to fix the first segment of track and again move the cursor. The second segment has not been fixed, but has acted as a guide to where the next section of track will be placed. <input type="checkbox"/> Press [F] to see the effect of the ASSIST FLIP command on the position of the 45 degree angle.

ANTICLOCKWISE Designator

Purpose	Rotates the current symbol designator anticlockwise
Mode	Symbol
Description	<p>Although the default position of a component designator is specified when the symbol is designed in the library editor, this command allows you to change its orientation while in the PCB editor. It rotates the designator 90 degrees anticlockwise about the centre of the first character.</p> <p>When you rotate a symbol, the designator is not normally affected since it is usual to have all the component designators facing the same way. This command allows you change this situation if required.</p>
Keys	[N]
Menu	{Option}{Designator ...}{Anticlockwise}
See also	TEXT CONTENT, EDIT SYMBOL, POSITION, HIDE/SHOW DESIGNATOR, SIZE, WIDTH
Example	<input type="checkbox"/> Load the sample board 'PLACED.PCB' into the PCB editor <input type="checkbox"/> Move the cursor near one of the symbols and press [F7]-(XT) or [F4]-(AT) to edit it <input type="checkbox"/> Press [N] and the designator will be rotated through 90 degrees anticlockwise <input type="checkbox"/> Press [N] three more times and it will be rotated back to its original orientation

ANTICLOCKWISE Designator prefix

Purpose	Rotates the designator prefix anticlockwise.
Mode	Desig
Requirements	You must be in the PCB library editor
Description	This command allows the user to determine the default angle at which the designator prefix is positioned in increments of 90 degrees. The prefix is rotated about the centre of the first character.
Keys	[A]
Menu	{Option}{Anticlockwise}
See also	MIRROR, SIZE, WIDTH, TEXTUAL CONTENT

ANTICLOCKWISE Text

Purpose	Rotates text anticlockwise.
Mode	Text
Description	This command allows the user to determine the angle at which text is orientated in increments of 90 degrees. The text is rotated about the centre of the first character in the current text string.
Keys	[A]
Menu	{Option}{Anticlockwise}
See also	ADD TEXT, SIZE, WIDTH, TEXTUAL CONTENT
Example	<ul style="list-style-type: none"><input type="checkbox"/> Select Add Text [F6]-(XT) or [F7]-(AT).<input type="checkbox"/> Type in a text string e.g. 'Twas brillig and the slithy toves' [Return].<input type="checkbox"/> Type [A] to rotate the text to the desired angle.<input type="checkbox"/> Press [Esc] to fix the text in position.

ANTICLOCKWISE Symbol

Purpose	Rotates a symbol anticlockwise.
Mode	Symbol
Requirements	There must be a current symbol selected.
Description	The orientation of a symbol which has been loaded from the symbol library is the same as it was when it was first created in the symbol library editor. It is possible to rotate symbols in increments of 90 degrees anticlockwise, giving the flexibility of creating only one symbol which can be used in any one of four orientations.
Keys	[A]
Menu	{Option}{Anticlockwise}
See also	ADD SYMBOL, MIRROR
Example	<input type="checkbox"/> Select Add Symbol by pressing [F8]. <input type="checkbox"/> When prompted for the symbol name type in 'DIP14' - this represents a 14 pin dual- in-line package. <input type="checkbox"/> At the symbol library filename prompt type in 'SYMBOL.LL' (if it has not already been given as the default library name) and press [Return]. This symbol or in this case component will now be displayed on the screen and can be rotated by pressing [A]. <input type="checkbox"/> When the component is at the desired angle press [Esc] to fix it in position.



ANTICLOCKWISE Pad

Purpose	Rotates pads anticlockwise.
Mode	Pad
Description	When placing pads on a board it is sometimes necessary to rotate them into different positions. Although this has little meaning when the pads are circular, some of the pad shapes provided in BOARDMAKER are rectangular, such as the edge connector fingers and may be rotated anticlockwise into one of four positions.
Keys	[A]
Menu	{Option}{Anticlockwise}
See also	ADD PAD, SHAPE, WIDTH, TRACK AND PAD SIZES
Example	<input type="checkbox"/> Select Add pad by pressing [F4]-(XT) or [F6]-(AT). <input type="checkbox"/> Select a pad SHAPE by pressing [S] and choose the edge connector finger L, from the list of shapes presented in the menu. <input type="checkbox"/> Rotate the pad anticlockwise by pressing [A]. <input type="checkbox"/> Press [Esc] to fix it in position.

ANTICLOCKWISE Block

Purpose Rotates a block and its contents anticlockwise.

Mode Block

Requirements The block must not be empty.

Description This allows a block which has been put around a group of items to be rotated with its contents about the current cursor position in increments of 90 degrees anticlockwise.

To determine which items in the block are picked up when the block is rotated and which are left behind, examine the settings using {Config}{Block options}.

It is possible using {Block options} to either pickup or leave tracks, pads, text or symbols as desired. Also, the tracks can be made to rubberband which will maintain the connectivity of any tracks you have placed between item inside the block and items outside the block.

Note that it is not possible to change the block settings if a block has been selected although it is possible to see what they are currently set to. If you need to change the block pickup settings, do it before you select the block or if a block is already selected temporarily drop it by pressing [Esc]. For more details on the block pickup options see CAPTURE MODE.

Keys [A]

Menu {Option}{Anticlockwise}

See also CLOCKWISE, CAPTURE MODE, BLOCK ORIGIN

Example

- ☐ Check to see the block options are set to track pickup and pad pickup on the {Config} menu. Do this by pressing [Ctrl][C] and selecting {Block options} from the {Config} options menu.
- ☐ Click on the Track and Pad menu options so they show as Track PICKUP and Pad PICKUP.

- ☐ Select CAPTURE MODE and select the option RUBBERBANDING OFF.
- ☐ Press [Esc] twice to return to the drawing screen.
- ☐ Place a number of pads and tracks close together. (See Add track and Add pad if you are not sure how to do this).
- ☐ Put the cursor slightly above and to the left of the group of tracks and pads and select block mode by pressing [F10]. This point is the first corner of the block and will be the origin.
- ☐ Move the cursor down to slightly below and to the right of the group, which will drag the block box over the items and press [Return].
- ☐ Press [A] and the tracks and pads contained in the box will be rotated anticlockwise by 90 degrees about the cursor position.
- ☐ Press [Esc] to fix the position of the components and drop the block.

ARC / CIRCLE

Purpose	Draws arcs and circles
Mode	Track
Requirements	The track must have exactly 2 nodes, the first is the centre of the circle, the second is a point on the circumference.
Description	<p>BOARDMAKER draws circles by expanding a track segment which has been placed between two points into a number of track segments which are arranged in a circle. This gives a great deal of flexibility in allowing the user to delete or insert sections of the circle or insert further track segments to construct arcs.</p> <p>The circle created is drawn on the same layer and with the same width parameters as the initial track segment presented, although any of the track segments can be switched to another layer if required (see LAYER). To construct an arc, simply remove those segments not required using DELETE.</p> <p>The number of track segments used to make the circle is determined by the user - although this can be set to auto tolerance which selects an appropriate number determined by the size of the circle. A low resolution circle contains approximately 30 nodes, a medium resolution circle contains approximately 55 nodes, and a high resolution circle contains approximately 80 nodes.</p>
Keys	[A]
Menu	{Option}{Arc/Circle}
See also	ADD TRACK, LAYER, AUTOVIA, WIDTH, TRACK AND PAD SIZES, HOME, END, BACK, NEXT, INVERT ORDER, DELETE
Example	<ul style="list-style-type: none"><input type="checkbox"/> Position the cursor where you want the centre of the circle to be and select Add Track [F2]-(XT) or [F5]-(AT).<input type="checkbox"/> Move the cursor to the right up to the point which you want the circle to pass through and press [Return]. Make

sure you only have only placed two nodes by using the information banner.

- ☐ Type [A] to select the Arc/Circle command.
- ☐ Press [Return] to select auto tolerance and the line will expand into a circle with its centre at the first point and passing through the second point.
- ☐ Press [Esc] to fix the circle in position.

ASSIST FLIP

Purpose To flip the position of the corner of 45 or 90 degree rubberbanded track segments.

Mode Track

Requirements This is only relevant when angle assist is set to 45 or 90 degrees.

Description When placing tracks with ANGLE ASSIST set to 45 or 90 degrees, BOARDMAKER automatically introduces a corner of either of these two angles so acting as a guide to the positioning of the track before actually fixing it in position. Since it is possible to have this corner in one of two positions the ASSIST FLIP command allows you to flip it between the two positions.

Keys [F]

Menu {Option}{Assist flip}

See also ANGLE ASSIST, CURRENT, MOVE NODE

- Example**
- ☐ Select Add track by pressing [F2]-(XT) or [F5]-(AT)
 - ☐ Select an ANGLE ASSIST of 45 degrees by pressing [Shift][A] until the status banner displays 45 degrees.
 - ☐ Start laying a track by moving the cursor and notice how a 45 degree corner is introduced.
 - ☐ Select ASSIST FLIP by pressing [F] and the position of the corner will flip into its alternate position.

AUTO PAN

Purpose To automatically pan the display

Mode All

Description It is possible to configure BOARDMAKER so that the editors will automatically pan by half a screen when the cursor touches the edge of the screen. This feature can be toggled on and off from the {Tools}{Auto pan} menu or by the direct keystroke [Shift][P].

Keys [Shift][P]

Menu {Tools}{Auto pan}

AUTO VIA

Purpose	To switch the auto via function
Mode	All
Description	<p>When you are laying a track and need to switch to another layer, you can arrange that BOARDMAKER will automatically put a via hole at the point where you switch layers. Three AUTO VIA options are available :</p> <ul style="list-style-type: none"><input type="checkbox"/> OFF - no via will be placed<input type="checkbox"/> ON - you will be prompted to accept the via placement<input type="checkbox"/> NO CONFIRM - a via will be placed and the layer changed with no confirmation <p>The pad size placed for this via is set from the TRACK AND PAD SIZES screen. Pad size number 16 is exclusively assigned as the via hole size and can be changed to suit your needs.</p> <p>If you have set up the layer pairs and switch AUTO VIA to NO CONFIRM, you simply press [Space] to immediately change to a new layer without BOARDMAKER prompting for the via.</p>
Keys	[Ctrl][C]{Auto via}
Menu	{Config}{Layer\Via setup}{Auto via}
See also	SEGMENT LAYER, ALL LAYERS, ALL WIDTH, SEGMENT WIDTH, LAYER PAIRS

BACK NET

Purpose	To display the previous un-routed net
Mode	All
Description	<p>This function allows you to display the previous un-routed net and hide all the others. It provides a fast and simple method of displaying the next set of connections you want to work on. Since only the points you need to connect are displayed, you do not have constantly refer back to your schematic diagram. This means you can quickly and accurately route your board.</p> <p>See the reference to NEXT NET for a full description and example</p>
Keys	[Shift][Tab]
Menu	{Net}{Back net}
See also	NEXT NET, NET HIGHLIGHT, DESIGN RULE CHECK, FULL DESIGN RULE CHECK

BACK Net node

Purpose	Moves the cursor to the previous node on the net
Mode	Net
Description	As nodes are placed to create a net they are numbered sequentially from the start to the end of the net. When editing a net, BACK causes the cursor to move the previous node on the net (panning if necessary), unless the cursor is initially at the first node, in which case nothing will happen.
Keys	[B]
Menu	{Option}{Back a node}
See also	EDIT NET, DELETE, END, HOME, NEXT, NET NAME, OPTIMIZE, CURRENT
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Add a new symbol pressing [F8] and enter 'DIP14' at the prompt. <input type="checkbox"/> Select ADD NET [Shift][F2]-(XT) or [Shift][F5]-(AT), enter the net name as 'N00001' [Return] and click on 4 of the symbol pins to make up a net of four nodes joined together with 3 lines. <input type="checkbox"/> Move BACK to the previous node by pressing [B]. <input type="checkbox"/> The cursor will jump to the previous node on the net.

BACK Track node

Purpose	Moves the cursor back one node along a track.
Requirements	The track must have more than one node.
Description	When a track is placed in position, it is made up of a series of track segments with a node at either end of each segment. These nodes are numbered sequentially from the start to the end of the track. When editing a track, BACK causes the cursor to move back to the preceding node (panning if necessary), unless the cursor is initially at the first or home node, in which case nothing will happen. The information banner displays the total number nodes in the track and the node which the cursor is currently on. This is shown in the form 'At node 3 of 12' indicating that the cursor is on node 3 of a track comprising 12 nodes, i.e 11 track segments.
Keys	[B]
Menu	{Option}{Back a node}
See also	HOME, END, NEXT, INVERT ORDER, DELETE, INSERT NODE, MOVE NODE
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Select Add track by pressing [F2]-(XT) or [F5]-(AT). <input type="checkbox"/> Move the cursor to create a new track segment and press [Return] to fix it in position. <input type="checkbox"/> Repeat this several times to create a number of track segments and note that the information banner shows the number of nodes you have created. <input type="checkbox"/> Press [B] to select the back command and the cursor will move back to the previous node. If this is repeated, the cursor can be moved back to the start of the track.

BACKUP TIMER

Purpose	To set the backup reminder alarm
Mode	All
Description	<p>BOARDMAKER provides the facility to remind you when you have been using the editor for pre defined period without doing a SAVE. This is a vital alarm which avoids you losing important work due to a power failure or hardware problem. The alarm (which you can turn off) can be set in the range 5 to 30 minutes in increments of 5 minutes. When the alarm triggers the status banner will display 'Save new version. Confirm Y/N' which gives you the choice to save a new version if required. If you do decide to save it, you will be warned that the file already exists before it is overwritten - exactly like a normal SAVE procedure.</p> <p><i>If you do not confirm a backup, you will be nagged every 5 minutes (regardless of the current setting) until you confirm a save. The backup timer will then revert to its set value.</i></p> <p>This procedure ensures that you can always go back to the last version you saved. If you make a severe error and need to use the backup copy, rename the .PC1 or .SC1 file to be a .PCB or .SCH file using the DOS copy command.</p>
Keys	[Ctrl][C]{Backup timer}
Menu	{Config}{Backup timer}
See also	SAVE, SAVE OPTIONS, CONFIG AUTO SAVE, MESSAGE TIMER

BLOCK

Purpose	To select block mode
Mode	All
Description	<p>This allows you to perform block related operations. A block is a section of workspace that can be marked and then manipulated as a single entity. Press [F10], move the block cursor to enclose the items you want and press [Return] to fix it. Use {Config}{Block options} to tell BOARDMAKER which items you wish to pickup or leave prior to selecting the block. The information banner will give you a summary of the items in the selected block.</p>
Keys	[F10]
Menu	{Tools}{Block mode}
See also	ANTICLOCKWISE, BLOCK OPTIONS, CLOCKWISE, BLOCK ORIGIN, KILL, MIRROR, REPEAT, SHIFT, FLOOD FILL

BLOCK OPTIONS

Purpose	To select which items are picked up by a block
Mode	Block
Description	When a block has been placed around a group of items on the drawing, this facility selects which items will be affected by subsequent block commands. You can choose to pickup or leave tracks, pads, text or symbols in any combination. You could, for example, put a block around a complete PCB layout which uses surface mount components and choose to mirror only the symbols and text onto the other side of the board. Consequently, the tracks and pads enclosed by the block would not be affected.

The BLOCK OPTIONS menu is available from the {Config} menu and displays the current settings for each drawing item as either PICKUP or LEAVE. To change the status for any item, move the menu banner over the item name in the menu and press [Return].

Also in this menu is the CAPTURE MODE option. The status of the capture mode determines whether tracks which pass through the block will be rubberbanded when the block is shifted or rotated. When set to rubberbanding on, a block operation will maintain connectivity. When set to rubberbanding off, only the tracks which are completely enclosed within the block will be affected.

Note that it is not possible to change the block settings if a block has been selected although it is possible to see what they are currently set to. If you need to change the block pickup settings, do it before you select the block or if a block is already selected temporarily drop it by pressing [Esc].

Keys	[Ctrl][C]{Block options}
Menu	{Config}{Block options}
See also	ANTICLOCKWISE, CLOCKWISE, BLOCK ORIGIN, KILL, MIRROR, REPEAT, SHIFT, FLOOD FILL

BLOCK ORIGIN

Purpose	To set the origin of a block
Mode	Block
Description	When a block has been placed around a group of items on a drawing, the origin of the block by default is the first corner. This origin is then used as a reference point for the block commands such as REPEAT and SHIFT. It is the point in the block which moves to the current cursor position.

It is sometimes convenient to be able to reposition the origin of the block to somewhere other than one of the block corners. BLOCK ORIGIN is designed to allow you to set a new origin for the block anywhere in the available drawing area - this can be either inside or outside the block.

This command is particularly useful when you want to position an item enclosed in the block exactly on another item somewhere else on the board.

Keys	[B]
Menu	{Option}{Block origin}
See also	ANTICLOCKWISE, CLOCKWISE, MIRROR, REPEAT, SHIFT, KILL
Example	<ul style="list-style-type: none"> <input type="checkbox"/> First make sure that the BLOCK OPTIONS are set to pickup pads and the CAPTURE MODE is set to rubberbanding off. Do this by selecting the {Block options} option from the {Config} menu - press [Ctrl][C] to directly get to the {Config} screen. <input type="checkbox"/> Select Add pad by pressing [F4]-(XT) or [F6]-(AT) and place a few pads by pressing [R] to repeat. Finally, press [Esc] to fix the last pad. <input type="checkbox"/> Select Block mode by pressing [F10] and drag the block box to completely enclose the pads you have placed.

- ☐ Move the cursor to a new position and select SHIFT by pressing [S]. The block and pads will be moved with the origin set to the current cursor position.
- ☐ Now put the cursor on one of the pads in the block and select BLOCK ORIGIN by pressing [B]. BOARDMAKER will beep to confirm the action.
- ☐ Again, move the cursor to a new position and select SHIFT by pressing [S] and the block and pads will be moved relative to the new origin.

BROWSE LIBRARIES

Purpose	To graphically browse the symbol libraries
Mode	All
Description	<p>The library browse option allows you to examine graphically the symbols in a library. You can scan through the library both forwards and backwards or inspect a particular symbol.</p> <p>Choose {Library}{Browse libraries} and the option menu will present you with the names of the symbols in the currently selected library; at the bottom of the list, the names of any other libraries will be shown. Move the banner over the name of the symbol you want to see and press [Return]. The screen will clear - you will not loose your work - and the symbol will be presented graphically with a small white cross indicating its origin. The name of the symbol will be displayed in the status banner along with the two options BACK and NEXT. Pressing [B] will display the previous symbol and [N] will display the next symbol. By pressing [Esc] you can get back to the option menu first displayed. To examine a different library, move the banner over its name in the option menu and press [Return] and the contents of that library will then be displayed.</p>
Keys	[Ctrl]{Browse libraries}
Menu	{Library}{Browse libraries}
See also	VIEW LIBRARY
Errors	<p>> No match found</p> <p>No symbol library could be found in the current library directory</p>

CLEAR NETS

Purpose	To remove all the nest from the PCB
Mode	All
Description	<p>This command provides a method of removing all the nets from the currently loaded PCB file - any tracks, pads text and symbols will be un-affected by the operation. You may find this function useful when you have partially created a net list and want to start again or if you want to explicitly clear a net list prior to using CREATE NETS.</p> <p>Use {Net}{Clear nets} to select the operation and since you could loose a great deal of information you will be prompted</p> <p>> All nets will be lost. Proceed (Y/N) ?</p>
Keys	[Ctrl][N]{Clear nets}
Menu	{Net}{Clear nets}
See also	LOAD NETS, EXPORT NETS, CREATE NETS, FULL DRC

CLEAR VIOLATIONS

Purpose	To clear the list of design rule violations
Mode	All
Description	<p>This command allows you to clear the current list of design rule violations produced by a DESIGNRULE CHECK. This list remains active until it is over- written by another check or it is explicitly cleared using this command.</p>
Keys	[Ctrl][T]{Clear violations}
Menu	{Tools}{Clear violations}
See also	DESIGN RULES, DESIGN RULE CHECK, NET HIGHLIGHT, NEXT VIOLATION, PREVIOUS VIOLATION, FULL DRC

CLOCKWISE Block

Purpose Rotates a block and its contents clockwise.

Mode Block

Requirements The block must not be empty.

Description This allows a block which has been put around a group of items to be rotated with its contents about the cursor position in increments of 90 degrees clockwise. To determine which items in the block are picked up when the block is rotated and which are left behind, examine the settings in {Config}{Block options}. It is possible using this option to either pickup or leave tracks, pads, text or symbols as desired. Also, the tracks can be made to rubberband which will maintain the connectivity of any tracks.

Note that it is not possible to change the block settings if a block has been selected although its is possible to see what they are currently set to. If you need to change the block pickup settings, do it before you select the block or if a block is already selected temporarily drop it by pressing [Esc]. For more details on the block pickup options see BLOCK OPTIONS.

Keys [C]

Menu {Option}{Clockwise}

See also ANTICLOCKWISE, BLOCK OPTIONS

Example

- ☐ Check to see the block options are set to track pickup and pad pickup on the {Config} screen. Do this by pressing [Ctrl][C] and selecting Block options from the main options menu.
- ☐ Click on the Track and Pad menu options so they show as Track PICKUP and Pad PICKUP.
- ☐ Press [Esc] twice to return to the drawing screen.
- ☐ Place a number of pads a tracks close together. (See Add track and Add pad if you are not sure how to do this).

- ☐ Put the cursor slightly above and to the left of the group of tracks and pads and select block mode by pressing [F10].
- ☐ Move the cursor down and to the right of the group, which will drag the block box over the items and press [Return].
- ☐ Move the cursor into the middle of the block.
- ☐ Press [C] and the tracks and pads contained in the box will be rotated clockwise by 90 degrees about the current cursor.
- ☐ Press [Esc] to fix the position of the items and drop the block.

CONNECT MODE

Purpose	To define the connect mode of a symbol pin
Mode	Ref
Requirements	You must be in the PCB library editor
Description	<p>This command provides the facility to explicitly determine the way a pad in a symbol is to be connected when in the PCB editor. When you are designing the symbol, you are able to set the pin attributes to be :</p> <p>Connect mode = ALWAYS</p> <p>The DRC will flag an error if you don't connect something to the pad</p> <p>Connect mode = NEVER</p> <p>The DRC will flag an error if you do connect something to the pad</p> <p>Connect mode = DON'T CARE</p> <p>The DRC will not care whether or not the pad is connected</p> <p>To set the connect mode, you first need to select the relevant reference by editing it {Edit}{Reference} and then selecting {Option}{Connect mode}. The {Option} menu will then present the three modes described above - simply select the one you require. Since the default setting is DON'T CARE, the reference box will not display the connect mode if this is selected.</p> <p>For a full description of references, see ADD REFERENCE</p>
Keys	[M]
Menu	{Option}{Connect mode}
See also	EDIT REFERENCE, NAME, ALIAS, ADD REFERENCE, FORCE LAYER

CREATE NETS

Purpose	To create a netlist for the entire layout
Mode	All
Description	<p>If you have a routed PCB without a netlist and want to create a full netlist from it, use this command. This has particular use when you design a layout manually and have not imported or created a netlist. You can then export this netlist and obtain a printed copy in order to manually check the connections. If you plan to use a BOARDMAKER in this way, you will find the checking process far easier if you create a netlist and check it rather than generate a check print and compare it against your schematic.</p> <p><i>Make sure your components have references.</i></p> <p>When you select {Net}{Create nets} you will be warned</p> <p>> All nets will be lost. Proceed (Y/N) ?</p> <p>if you already have any nets. Press [Y] if it is okay to loose the existing nets and you will then see the message</p> <p>> Process may take some time. Please wait.</p> <p>This tells you that BOARDMAKER is making the netlist and you will see each connection being highlighted. When finished, all the nets created will be automatically displayed.</p>
Keys	[Ctrl][N]{Create nets}
Menu	{Net}{Create nets}
See also	LOAD NETS, EXPORT NETS, FULL DRC

CURRENT Designator prefix

Purpose Returns the cursor to the current designator.

Mode Desig

Requirements You must be in the PCB library editor

Description If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current designator. If this involves panning back to the original screen, then this will be performed automatically.

Keys [C]

Menu {Option}{Current}

See also ZOOM, PAN, UNZOOM

Example

- ☐ Put the cursor in the lower left corner of the screen of the PCB library editor.
- ☐ Select Add designator [Shift][F6]-(XT) or [Shift][F7]-(AT) and type in some text. e.g TR [Return]
- ☐ Move the cursor to the upper right edge of the screen and press [P] to pan to a different area of the board.
- ☐ Press [C] and the display will pan back and position the cursor on the designator just placed - the current designator.

CURRENT Net

Purpose Returns the cursor to the current net node.

Mode Net

Requirements You must be in the PCB editor

Description During net adding or editing, one of the nodes will always be considered to be the current node. If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current node. If this involves panning back to the original screen, then this will be performed automatically.

Keys [C]

Menu {Option}{Current}

See also ZOOM, PAN, UNZOOM

Example

- ☐ Add a new symbol pressing [F8] and enter 'DIP14' at the prompt and enter the library name as 'SYMBOL.LL'.
- ☐ Select ADD NET [Shift][F2]-(XT) or [Shift][F5]-(AT), enter the net name as 'N00001' [Return] and click on several of the symbol pins to make up a net.
- ☐ Move the cursor to the edge of the screen and press [P] to pan to a different area of the board.
- ☐ Press [C] and the display will pan back and position the cursor on the last node which was placed - the current node.

CURRENT Pad

Purpose	Returns the cursor to the current pad.
Mode	Pad
Description	During pad placement or editing, the pad last placed will be considered to be the current pad. If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current pad. If this involves panning back to the original screen, then this will be performed automatically.
Keys	[C]
Menu	{Option}{Current}
See also	ZOOM, PAN, UNZOOM
Example	<input type="checkbox"/> Select Add pad [F4]-(XT) or [F6]-(AT) to create a new pad <input type="checkbox"/> Move the cursor to the edge of the screen and press [P] to pan to a different area of the board. <input type="checkbox"/> Press [C] and the display will pan back and position the cursor on the pad just placed - the current pad.

CURRENT Reference

Purpose	Returns the cursor to the current reference
Mode	Ref
Requirements	You must be in the PCB library editor
Description	When you are designing a symbol and are adding or editing references, the last once placed is considered to be the CURRENT reference. If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current reference. If this involves panning back to the original screen, then this will be performed automatically.
	For a full description of references, see ADD REFERENCE
Keys	[C]
Menu	{Option}{Current}

CURRENT Symbol

Purpose	Returns the cursor to the current symbol.
Mode	Symbol
Description	During symbol placement, the last symbol placed will be considered to be the current one. If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current symbol. If this involves panning back to the original screen, then this will be performed automatically.
Keys	[C]
Menu	{Option}{Current}
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Select Add symbol [F8]. <input type="checkbox"/> When prompted for the symbol name enter 'DIP14' [Return]. <input type="checkbox"/> Enter 'SYMBOL.LL' for the symbol library name if prompted. This recalls a symbol for a 14 pin IC from the standard library. <input type="checkbox"/> Move the cursor to the edge of the screen and press [P] to pan to a different area of the board. <input type="checkbox"/> Press [C] and the display will pan back and position the cursor on the symbol just placed - the current symbol.

CURRENT Text

Purpose	Returns the cursor to the current text.
Mode	Text
Description	During text placement or editing, the last text placed will be considered to be the current text. If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current text. If this involves panning back to the original screen, then this will be performed automatically.
Keys	[C]
Menu	{Option}{Current}
See also	ZOOM, PAN, UNZOOM
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Put the cursor in the lower left corner of the screen. <input type="checkbox"/> Select Add text [F6]-(XT) or [F7]-(AT) and type in some text. <input type="checkbox"/> Move the cursor to the upper right edge of the screen and press [P] to pan to a different area of the board. <input type="checkbox"/> Press [C] and the display will pan back and position the cursor on the text just placed - the current text.

CURRENT Track

Purpose	Returns the cursor to the current node.
Mode	Track
Description	During track placement or editing, one of the nodes will always be considered to be the current node and the track segment between the current node and the subsequent node is called the current track segment. If the cursor is then moved away to a different area of the screen, for example by panning, then this single keystroke command can be used to return the cursor to the current node. If this involves panning back to the original screen, then this will be performed automatically.
Keys	[C]
Menu	{Option}{Current}
See also	ZOOM, PAN, UNZOOM
Example	<div><input type="checkbox"/> Select Add track [F2]-(XT) or [F5]-(AT) and place several track segments on the screen.</div> <div><input type="checkbox"/> Move the cursor to the edge of the screen and press [P] to pan to a different area of the board.</div> <div><input type="checkbox"/> Press [C] and the display will pan back and position the cursor on the last node which was placed - the current node.</div>

CURSOR BOX

Purpose	To switch the mode selected indicator on and off.
Mode	All
Description	Along with the mode indicator in the middle of the status banner and the display in the information banner, it is possible to show that a mode is selected by turning the cursor box on. When a mode is then selected, the cursor will be surrounded by a small box. This is an optional facility which is enabled or disabled in the {Config}{Cursor settings} menu - clicking on the {Cursor box} option will turn the box ON or OFF.
Keys	None
Menu	{Config}{Cursor settings}{Cursor box}
See also	CURSOR SIZE, CURSOR MODE, CURSOR TYPE
Example	<div><input type="checkbox"/> From the PCB editor select {Config}{Cursor settings}</div> <div><input type="checkbox"/> Click on the {Cursor box} option until it shows {Cursor box ON}</div> <div><input type="checkbox"/> Press [Esc] twice to get back to the PCB editor. Note that the cursor is simply a cross hair.</div> <div><input type="checkbox"/> Select {Add}{Track} and the cursor will be surrounded by a small box and the status banner will display 'Track', both indicating that track mode is active.</div>

CURSOR MODE

Purpose	To switch the cursor mode.
Mode	All
Description	BOARDMAKER provides two cursor modes, one of which is absolute, the other is relative. If the cursor mode is set to absolute, then the cursor readout on the status banner is the X and Y distances from the bottom left hand corner of the available drawing area, the fixed origin. If the cursor mode is relative, the cursor readout is the X and Y distance from a floating origin set by the user. The status banner always displays the current cursor mode with an 'A' for absolute or an 'R' for relative. This command allows you to toggle the cursor mode between absolute and relative.
Keys	[Shift][M]
Menu	{Tools}{Cursor mode}
See also	MOUSE LOCKING, SET ORIGIN, GOTO ORIGIN, UNITS
Example	<div><input type="checkbox"/> Put the cursor somewhere in the middle of the drawing area.</div> <div><input type="checkbox"/> Select GOTO ORIGIN by pressing [Shift][O] and the cursor will move to the bottom left hand corner - the default floating origin.</div> <div><input type="checkbox"/> Now move the cursor back to the middle of the drawing area and set the floating origin by pressing [O]. The origin is now set to the current cursor position but the cursor readout on the status banner is still in absolute mode, indicated by the 'A'.</div> <div><input type="checkbox"/> Change the CURSOR MODE to relative by pressing [Shift][M] and provided the cursor has not moved, the cursor readout should now display 0,0. You have now set a new floating origin.</div>

CURSOR SIZE

Purpose	To change the size of the target cursor
Mode	All
Description	This command provides a method of changing the size of the target cursor and has three settings - small, medium, or large. To select a new cursor size, select Cursor size from the {Config} menu and you will be presented with the three size options. Simply move the menu banner over the one you want and press [Return].
Keys	[Ctrl][C]{Cursor settings}{Cursor size}
Menu	{Config}{Cursor settings}{Cursor size}
See also	MOUSE LOCKING, CURSOR TYPE, CURSOR MODE, CURSOR TRAVEL, CURSOR BOX

CURSOR TRAVEL

Purpose	To change the way the cursor moves.
Mode	All
Description	Using this command you are able to make the cursor either snap to the snap grid selected or smoothly glide around the screen. This action can either be permanently set using the [NumLock] key or temporarily set by holding the [Alt] and any combination of arrow keys simultaneously. This is useful if you do not have a mouse attached to your computer.
Keys	[Alt] + arrow key to temporarily glide the cursor or [NumLock] to switch between snap and glide. The cursor will glide when the Num Lock light is on.
See also	MOUSE LOCKING, CURSOR TYPE, CURSOR SIZE, GRID SNAP, GRID

CURSOR TYPE

Purpose	To switch the cursor shape between the target type and the cross hair type
Mode	All
Description	This command allows you to use one of the two types of cursor available in BOARDMAKER. The target cursor, which is the default cursor type, is a small cross hair which can be changed in size using the CURSOR SIZE option in {Config}{Cursor settings}. The cross hair cursor is a full width cursor which extends to the edges of the screen and is provided for instances when items placed on the board need to be lined up with adjacent items.
Keys	[Shift][C]
Menu	{Tools}{Cursor type}
See also	MOUSE LOCKING, CURSOR SIZE
Example	<div><input type="checkbox"/> Select Add track by pressing [F2]-(XT) or [F5]-(AT). The cursor displayed is a medium size target, the default setting.</div> <div><input type="checkbox"/> Select CURSOR TYPE by pressing [Shift][C] and the cursor will change to a full width cross hair.</div>

DECREMENT Alias

Purpose	To decrement the reference alias
Mode	Ref
Requirements	You must be in the PCB library editor
Description	<p>This command allows you to decrement characters and numbers in the reference alias. When designing a symbol which uses these, you can use the REPEAT command to quickly copy references you have already placed to a new position complete with name and alias. DECREMENT allows you to easily alter the alias if it is similar. For example, if it is currently 'DATA15' it will decrement to 'DATA14', and so on. To do this, you must first select the reference, if it is not already selected, by editing it {Edit}{Reference} and then selecting {Option}{Decrement alias} will decrement it. This greatly speeds up the process of creating symbols with similar references.</p> <p>If you try to decrement when the last character is 'A' or ends in a '0' you will get the error Increment/decrement failure.</p> <p>For a full description of references, see ADD REFERENCE</p>
Keys	[Shift][D]
Menu	{Option}{Decrement alias}
See also	EDIT REFERENCE, INCREMENT ALIAS, DECREMENT NAME, INCREMENT NAME

DECREMENT Name

Purpose	To decrement the reference name
Mode	Ref
Requirements	You must be in the PCB library editor
Description	<p>This command allows you to decrement characters and numbers in the reference name. When designing a symbol which uses these, you can use the REPEAT command to quickly copy references you have already placed to a new position complete with name and alias. DECREMENT allows you to easily alter the name if it is similar. For example, if a pin number on an IC is 10, it will decrement to 9, and so on. To do this, you must first select the reference, if it not already selected, by editing it {Edit}{Reference} and then select {Option}{Decrement name}. This greatly speeds up the process of creating symbols with similar references.</p> <p>If you try to increment when the last character is a 'Z' or 'z' you will get the error Increment/decrement failure.</p> <p>For a full description of references, see ADD REFERENCE</p>
Keys	[D]
Menu	{Option}{Decrement name}
See also	EDIT REFERENCE, INCREMENT ALIAS, INCREMENT NAME, DECREMENT ALIAS, ADD REFERENCE

DECREMENT Text

Purpose	To decrement the current text
Mode	Text
Description	<p>This command allows you to decrement characters and numbers in text. When placing numbers and letters, for example on connectors or IC pin numbers, it allows you to easily decrement the last character of the text string - if it is a number the whole number will increment. e.g. IC1000 becomes IC999, IC2c becomes IC2b, Conn-T becomes Conn-S. This can greatly speed up the process of numbering components, when used in conjunction with REPEAT.</p> <p>If you try to decrement when the last character is an 'A' or 'a' you will get the error Increment/decrement failure.</p>
Keys	[D]
Menu	{Tools}{Decrement}
See also	INCREMENT
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Select Add text by pressing [F6]-(XT) or [F7]-(AT) <input type="checkbox"/> Enter some text e.g.'IC1000' [Return] <input type="checkbox"/> Move the cursor to the right and REPEAT it by pressing [R]. You will now see IC1000 displayed twice. <input type="checkbox"/> Decrement the text by pressing [D] and you will now see the current text displayed as IC999

DELETE Net node

Purpose	Deletes a node from a net
Mode	Net
Description	<p>This command allows the user to delete selected nodes from a net. As a node is deleted the two net lines which are joined to it (if it is in the middle of a track) will be deleted and a new line will be drawn between the previous and the next node. This must be used with care and BOARDMAKER will ask you to confirm each deletion.</p>
Keys	[D]
Menu	{Option}{Delete}
See also	EDIT NET, HOME, END, BACK, NEXT, NET NAME, OPTIMIZE, CURRENT
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Add a new symbol pressing [F8] and enter 'DIP14' at the prompt. <input type="checkbox"/> Select ADD NET [Shift][F2]-(XT) or [Shift][F5]-(AT), enter the net name as 'N00001' [Return] and click on 4 of the symbol pins to make up a net of four nodes joined together with 3 lines. <input type="checkbox"/> Move BACK to node 2 by pressing [B] twice. <input type="checkbox"/> Delete node 2 by pressing [D]. You will be prompted to confirm - press [Y]. The net will now be displayed with only 3 nodes joined by 2 lines.

DELETE Track

Purpose	Deletes track segments
Mode	Track
Description	When a track is placed in position, it is made up of a series of track segments with a node at either end of each segment. These nodes are numbered sequentially from the start to the end of the track.

This command allows the user to delete track segments by deleting nodes along the track. As a node is deleted the two track segments which are joined to it (if it is in the middle of a track) will be deleted and a new segment will be drawn between the previous and the next node. If the current node is the end node, only one track segment will be deleted. If this is repeated, the track is effectively 'ripped up' from the end to the home node. To do this starting from the home end of the track, use the INVERT ORDER command to logically invert the order of the nodes on the track - this swaps the home and end nodes around, and then 'rip up' from the end node as before.

Keys	[D]
Menu	{Option}{Delete}
See also	SPLIT TRACK, HOME, END, NEXT, BACK, INVERT ORDER, KILL
Example	<ul style="list-style-type: none"><input type="checkbox"/> Select Add track by pressing [F2]-(XT) or [F5]-(AT).<input type="checkbox"/> Turn rubberbanding on by pressing [Shift][B] and place at least six track segments.<input type="checkbox"/> Press [D] for delete and the last track segment placed will be removed.<input type="checkbox"/> Move back to node three in the middle of the track by pressing [B]. Use the information banner to see which node you are on.<input type="checkbox"/> Press [D] to delete a node and notice how the two track segments on node three are deleted.

DESIGNATOR POSITION

Purpose	To change the position of a designator
Mode	Symbol
Description	Although the default position of a component designator is specified when the symbol is designed in the library editor, this command allows you to change its position while in the PCB editor. To move a symbol designator, you must first edit the symbol and then select {Option}{Designator ...} and the {Option} menu then presents you with the operations available on designators. Choose {Position} and you are prompted :

> Position cursor and press [Return]

Move the cursor where you want the designator to be and press [Return]. When you are editing a symbol and want to quickly move the designator, you can use a direct keystroke. Do this by holding the [Shift] key and pressing [Return] or the left hand mouse button.

Keys	[D]{Position} or [Shift][Return]
Menu	{Option}{Designator ...}{Position}
See also	TEXT CONTENT, EDIT SYMBOL, ANTICLOCKWISE, HIDE/SHOW DESIGNATOR, SIZE, WIDTH
Example	<ul style="list-style-type: none"><input type="checkbox"/> Load the sample board 'PLACED.PCB' into the PCB editor<input type="checkbox"/> Move the cursor near one of the symbols and press [F7]-(XT) or [F4]-(AT) to edit it<input type="checkbox"/> Move the cursor to a new position and press [Return]. Notice how the designator moves with the symbol.<input type="checkbox"/> Now press and hold [Shift] move the cursor and again press [Return]. This time, notice that only the designator moves and not the symbol.

DESIGN RULES Global

Purpose	To set the global design rules
Mode	All
Description	<p>This command allows you to define the minimum clearances or separation between particular items on a layout. These are the default design rules which are applied globally. i.e. the clearances set are the same regardless of the type of track you are placing. You set the design rules from {Config}{Design rules}, where you will be presented with the design rule check (DRC) menu. As well as allowing you choose which rules you want to set, it displays the current clearances. You have the option to set the track to track, track to pad or pad to pad minimum clearances. The last option on the menu allows you to determine whether the design rule checker is automatically started as soon as a net has been highlighted - by default, this is turned on.</p> <p>Select the rule you want to set and you will be presented with an edit window which allows you to change the setting. Use either the up or down arrow keys to increase or decrease the displayed value - if you have a mouse, you can do this by moving the mouse forwards and backwards respectively.</p> <p>You can set the clearance in the range 4 to 250 thou inclusive.</p>
Keys	[Ctrl][C]{Design rules}
Menu	{Config}{Design rules}
See also	DESIGN RULE CHECK, NET HIGHLIGHT, NEXT VIOLATION, PREVIOUS VIOLATION, CLEAR VIOLATIONS, FULL DRC

DESIGN RULES Net based

Purpose	To set the net based design rules
Mode	Net
Requirements	You must be in the PCB editor
Description	<p>This command allows you to define the minimum clearances or separations between particular items on a layout from a named net. When nets are imported or created, the design rules will all default to 10 thou. You are then able to change the design rules for a particular net to different values. This very powerful feature ensures that the DRC and route verification procedures can automatically perform a very accurate check of your layout in a similar manner to manual checking.</p> <p>For example, if you are designing a microprocessor based board with an on board mains power supply, you can use this facility to define different track clearances for the nets on the address bus to those on the mains input of the transformer primary.</p> <p>You set the clearances by first editing the required net and selecting {Option}{Design rules ...}. The {Option} menu will then prompt you to set the track to track, track to pad or pad to pad rules. Select the one you want to change and the line editor will typically display :</p> <p>> Track <-> Track clearance is : 10 thou</p> <p>To change this value, simply use the up and down arrows to increment it or decrement it.</p>
Keys	[R]
Menu	{Option}{Design rules ...}
See also	DESIGN RULE CHECK, NET HIGHLIGHT, NEXT VIOLATION, PREVIOUS VIOLATION, CLEAR VIOLATIONS, FULL DRC

DESIGN RULE CHECK (DRC)

Purpose To verify a net and the apply design rules

Mode Hilite

Description This command allows you to apply the design rules you have set to a highlighted net and is intended to aid the checking process when have designed your PCB layout. If there is an associated net, the DRC will validate the routed sections against the net and if correct, will flag them as routed. Consequently, when you use NEXT NET or BACK NET to show the next or previous un-routed net, only the outstanding routes will be displayed.

To apply the design rules to a net, you must have first highlighted the net. If you have enabled auto check in {Config}{Design rules}, the design rule check will automatically take place after the net highlighting has finished. The progress of the DRC is displayed on the status banner in the following format :

```
> Using design rules for net N00001
```

where N00001 is the name of the net being checked or if there are no symbols with references it will display :

```
> Using global design rules for DRC
```

If there are some DRC violations, the information banner will display it in the following format :

```
> T<->T:12 T<->P:23 P<->P:6
```

This tells you there are 12 track to track violations, 23 pad to pad violations and 6 pad to pad violations.

Warnings > Highlighted items on more than one net

If the highlighted net was shorted with another net, you will get this warning. You have probably routed the highlighted net incorrectly.

```
> Warning! Extra nodes on net
```

This tells you that the highlighted net has additional nodes on it which are not part the net. You may have incorrectly routed the net since it is touching something else on the layout.

```
> Text on net! Remove before proceeding
```

This happens if there is text on a copper layer which has been highlighted as part of the net. For DRC purposes, text is not permitted on a net, although it will be highlighted if it is connected. This ensures that any text on a copper layer which connects two nets can be identified.

See also

DESIGN RULES, NET HIGHLIGHT, NEXT VIOLATION, PREVIOUS VIOLATION, CLEAR VIOLATIONS

Example

- ☐ Highlight a net by positioning the cursor over a track or pad and press [F9].
- ☐ Perform a DESIGN RULE CHECK by pressing [D].
- ☐ You will be warned if there are any violations and if so, you can view each in turn using NEXT VIOLATION or PREVIOUS VIOLATION.
- ☐ If there are no violations you will get the message : No violations found.

DIRECTORY

Purpose	To display a directory of the artwork files in the working directory.
Mode	All
Description	<p>You will often need to know which layout and schematic files are saved on your disc. This command provides a quick and efficient method of keeping track of the artwork files in the current working directory. The filename mask is *.PCB for layout files and *.SCH for schematic files - this is fixed within BOARDMAKER and the mask will be automatically changed as you switch between schematic and layout mode.</p> <p>BOARDMAKER will search for the files in the current working directories. These can be viewed or changed using the DIRECTORY SETUP command in the {Config} menu. This facility allows you to set the PCB, Schematic, Library, Net and working directories in whatever way suits your needs.</p>
Keys	[Alt][D]
Menu	<p>{File}{Dir *.PCB} for layouts.</p> <p>{File}{Dir *.SCH} for schematics.</p>
See also	LOAD, SAVE, SAVE OPTIONS, CONFIG AUTO SAVE, BACKUP TIMER, NEW, DIRECTORY SETUP
Example	<p><input type="checkbox"/> Select directory by pressing [Alt][D] from within the layout editor.</p> <p><input type="checkbox"/> The {Option} menu will now show a list of all the PCB files in the current directory. The currently loaded file will be displayed in red.</p> <p><input type="checkbox"/> Press [Esc] to return to the main display.</p>

DIRECTORY SETUP

Purpose	To view or change BOARDMAKER's working directories
Mode	All
Description	<p>The entries presented in the menu when this command is selected show where BOARDMAKER looks to find the various files you wish to use. For details of which configuration options are saved see the SAVE OPTION command.</p> <p>Current directory displays the current DOS directory and allows you to change to a specified drive and directory.</p> <p>PCB directory displays the DOS directory where the PCB files will be loaded or saved. If the entry is blank, the files are loaded and saved in the current directory.</p> <p>Schematic directory displays the DOS directory where the Schematic files will be loaded or saved. If the entry is blank, the files are loaded and saved in the current directory.</p> <p>Library directory displays the DOS directory where the library files are loaded, saved or read. If the entry is blank, BOARDMAKER will look in the current directory.</p> <p>Net directory displays the DOS directory where the net and map files are loaded, saved or read. If the entry is blank, BOARDMAKER will look in the current directory.</p> <p>Before using this command, make sure you are fully conversant with the use of DOS drive and path directives. Use the following as a guide-line when entering directory paths:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Relative and absolute path names are allowed, including path names relative to the current directory. <input type="checkbox"/> Entries such as . (the current directory) .. (the parent directory) .. \ (a sibling directory) are all valid. <p>Example setups:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Current directory - C:\BM i.e. current DOS directory is C:\BM

- ☐ PCB directory - PCB i.e. actual path C:\BM\PCB
- ☐ Schematic directory - \BM\SCH i.e. actual path C:\BM\SCH
- ☐ Library directory - ..\LIBRARY i.e. actual path C:\LIBRARY

Keys [Ctrl][C]{Directory setup}

Menu {Config}{Directory setup}

See also SAVE, LOAD, SAVE OPTIONS, DIRECTORY, BROWSE LIBRARIES

EDIT DESIGNATOR

Purpose	To edit an existing designator prefix
Mode	All
Requirements	You must be in the PCB library editor
Description	This command provides a method of editing a designator prefix which has already been placed. What you are actually doing is selecting it prior to performing some other designator editing command such as changing the size or the textual content. When the designator is selected it becomes the CURRENT designator and as a consequence is mobile. If you move the cursor and press [Return] it will move to the cursor position.
Keys	[Shift][F5]-(XT) or [Shift][F3]-(AT) [Ctrl][E]{Designator}
Menu	{Edit}{Designator}
See also	EDIT REFERENCE, ANTICLOCKWISE, CURRENT, KILL, LAYER, AUTOVIA, MIRROR, REPEAT, SIZE, TEXT CONTENT, WIDTH

EDIT NET

Purpose To edit an existing net.

Mode Net

Description This command provides a method of editing nets which have already been added to a layout. What you are actually doing is selecting a net prior to performing some other editing command such as changing its name or deleting a node. When the net is selected it becomes the CURRENT net and the information banner will display :

```
> Net=N00001 Tot=3 (10,12,15)
```

indicating you are editing a net named N00001 which has three nodes and the design rules are set to be :

Track to track 10 thou

Track to pad 12 thou

Pad to pad 15 thou

The nodes are numbered sequentially from the start to the end of the net.

Since nets have a name, there are two ways of defining which net you want to edit - you can edit the net by selecting one of its nodes using {Edit}{Net} or by naming it using {Net}{Edit net}.

EDITING BY NAME

If you want to edit a net and you know its name but are unsure of its exact nodes, select {Net}{Edit net} and the {Option} menu will present you with the names of all the nets. When you select the one you need to edit, all other nets will be hidden and the selected net will be displayed. You are then able to modify the net as required.

EDITING BY NODE

If you want to edit a net and you know where one of its nodes is, simply put the cursor on the node and select EDIT NET

Keys [Shift][F1] to select net by node

[Ctrl][N]{Edit}{Net} to select net by name

Menu {Edit}{Net} to select net by node

{Net}{Edit net} to select net by name

See also ADD NET, BACK, CURRENT, DELETE, END, HOME, KILL, NEXT, NET NAME, OPTIMIZE

EDIT PAD

Purpose	To edit an existing pad.
Mode	All
Description	This command provides a method of editing pads which have already been placed on the drawing. What you are actually doing is selecting the pad prior to performing some other editing command such as changing its size or shape. When the pad is selected it becomes the CURRENT pad and as a consequence is mobile. If you move the cursor and press [Return] the pad will move to the new cursor position. To fix the pad in position press [Esc].
Keys	[F3]-(XT) or [F2]-(AT) [Ctrl][E]{Pad}
Menu	{Edit}{Pad}
See also	ANTICLOCKWISE, CURRENT, KILL, LAYER, AUTOVIA, MIRROR, REPEAT, SHAPE, WIDTH

EDIT REFERENCE

Purpose	To edit an existing pin reference
Mode	All
Requirements	You must be in the PCB library editor
Description	This command provides an method of editing an existing symbol reference. What you are actually doing is selecting a reference prior to performing some other reference editing command such as adding an alias or moving the reference box. When the reference is selected it becomes the CURRENT reference. For a full description of references see ADD REFERENCE.
Keys	[Shift][F3]
Menu	{Edit}{Reference}
See also	ADD DESIGNATOR, ADD REFERENCE, NAME, ALIAS, REFERENCE BOX, FORCE LAYER, CONNECT MODE, INCREMENT ALIAS, INCREMENT NAME

EDIT SYMBOL

Purpose	To edit an existing symbol.
Mode	All
Description	This command provides a method of editing symbols which have already been placed on the drawing. What you are actually doing is selecting the symbol prior to performing some other symbol command such as changing its position or duplicating it. When the symbol is selected it becomes the CURRENT symbol and as a consequence is mobile. If you move the cursor and press [Return] it will move to the new cursor position. To fix the symbol in position press [Esc].
Keys	[F7]-(XT) or [F4]-(AT) [Ctrl][E]{Symbol}
Menu	{Edit}{Symbol}
See also	DESIGNATOR, VALUE, ANTICLOCKWISE, CURRENT, KILL, MIRROR, REPEAT

EDIT TEXT

Purpose	To edit existing text.
Mode	All
Description	This command provides a method of editing text which has already been placed on the drawing. What you are actually doing is selecting the text prior to performing some other editing command such as changing its size or content. When the text is selected it becomes the CURRENT text and as a consequence is mobile. If you move the cursor and press [Return] the text will move to the new cursor position. To fix the text in position press [Esc].
Keys	[F5]-(XT) or [F3]-(AT) [Ctrl][E]{Text}
Menu	{Edit}{Text}
See also	ANTICLOCKWISE, CURRENT, KILL, LAYER, MIRROR, REPEAT, SIZE, TEXT CONTENT, WIDTH

EDIT TRACK

Purpose	To edit existing tracks or lines.
Mode	All
Description	<p>This command provides a method of editing tracks or lines which have already been placed on the drawing. What you are actually doing is selecting the track segment prior to performing some other track editing command such as changing the width or moving the track segment. When a track is placed in position, it is made up of a series of track segments with a node at each end. These nodes are numbered sequentially from the start to the end of the track.</p> <p>The current track segment is defined as the line joining the current node and the subsequent node and when selected is displayed on the screen with small sections missing at either end.</p>
Keys	<p>[F1]</p> <p>[Ctrl][E]{Track}</p>
Menu	{Edit}{Track}
See also	BACK, CURRENT, DELETE, END, ASSIST FLIP, HOME, INVERT ORDER, KILL, SEGMENT LAYER, ALL LAYERS, INSERT/MOVE NODE, NEXT, REPEAT, SPLIT, SHIFT, ALL WIDTH, SEGMENT WIDTH

END Net node

Purpose	Moves the cursor to the end node on the net
Mode	Net
Description	As nodes are placed to create a net they are numbered sequentially from the start to the end of the net. When editing a net, END causes the cursor to move the last node on the net (panning if necessary).
Keys	[E]
Menu	{Option}{End}
See also	EDIT NET, DELETE, HOME, BACK, NEXT, NET NAME, OPTIMIZE, CURRENT
Example	<p><input type="checkbox"/> Add a new symbol pressing [F8] and enter 'DIP14' at the prompt.</p> <p><input type="checkbox"/> Select ADD NET [Shift][F2]-(XT) or [Shift][F5]-(AT), enter the net name as 'N00001' [Return] and click on 4 of the symbol pins to make up a net of four nodes.</p> <p><input type="checkbox"/> Move to the home node by pressing [H].</p> <p><input type="checkbox"/> The cursor will jump to the first node you selected.</p> <p><input type="checkbox"/> Move the cursor to the end node by pressing [E]</p>

END Track node

Purpose	Moves the cursor to the last node of the track.
Mode	Track
Description	When a track is placed in position, it is made up of a series of track segments with a node at either end of each segment. These nodes are numbered sequentially from the start to the end of the track. When editing a track, END causes the cursor to move the last node placed (panning if necessary).
Keys	[E]
Menu	{Option}{End}
See also	HOME, BACK, NEXT, INVERT ORDER, DELETE
Example	<div><input type="checkbox"/> Select Add track by pressing [F2]-(XT) or [F5]-(AT).</div> <div><input type="checkbox"/> Move the cursor to create a new track segment and press [Return] to fix it in position.</div> <div><input type="checkbox"/> Repeat this several times to create a number of track segments and note that the information banner shows the number of nodes you have created.</div> <div><input type="checkbox"/> Move back along the track a number of nodes by pressing [B].</div> <div><input type="checkbox"/> Press [E] and observe the cursor moves to the last node you placed on this track i.e. the end node.</div>

ERASE SYMBOL

Purpose	Erases an item from a symbol library.
Mode	All
Requirements	You must be in the library editor
Description	As part of BOARDMAKER's library management facilities, this command provides a method of deleting any symbol from any library. You will be asked for the symbol name you wish to delete, the library name you wish to delete the symbol from and lastly confirmation that you actually want to go ahead and delete the symbol.
Keys	[Ctrl][L]{Erase symbol}
Menu	{File}{Symbol library...}{Erase symbol} for all displays or {Library}{Erase symbol} for EGA and VGA displays
See also	BROWSE LIBRARIES, ADD TO LIBRARY, VIEW LIBRARY, UPDATE SYMBOLS
Example	<div><input type="checkbox"/> Before following this example, first see the example in the ADD TO LIBRARY command and create a dummy symbol called 'TEMP'.</div> <div><input type="checkbox"/> Now select {Erase symbol} from the {Library} menu and you will be prompted for the symbol name - enter 'TEMP' [Return]</div> <div><input type="checkbox"/> Enter the symbol library name as 'SYMBOL.LL' [Return]</div> <div><input type="checkbox"/> When prompted to confirm the deletion press 'Y' and the symbol 'TEMP' will be deleted from the library.</div>

EXIT

Purpose	To quit and return to the main menu
Mode	All
Description	This command allows you to finish editing and return to the main BOARDMAKER menu. You will be asked to confirm the fact that you wish to quit and also if you have not saved the current drawing you will be warned. This ensures that you cannot leave the editor with a drawing you have not saved without knowing about it.
Keys	[Alt][X]
Menu	{File}{Exit editor}
See also	NEW, SAVE, SAVE OPTIONS, CONFIG AUTO SAVE, BACKUP TIMER

EXPORT NET

Purpose	To export a net to a file
Mode	All
Description	<p>All net information is held in the PCB file, consequently, if you want to see the netlist you need to export it from the PCB file to a named net file. Select {Net}{Export nets} and you will be presented with the Export net screen.</p> <p>The information window shows the name of the current PCB file and the net file name defaults to the same name, but with the file extension .NET. To create the net file, choose {Export net} and you are then prompted for the net file name.</p> <p>Press [Return] to select the default name or enter new file name and the net file will be created.</p> <p>The {Convert to ..} option allows you translate a BOARDMAKER net list to other net list formats. Select the format required and you are prompted for the BOARDMAKER net file name and the other format file name. When both are entered, the file will be translated to the desired format. Note that the original net file and PCB file will not be affected by this procedure.</p> <p>Select {Exit} to return to the PCB editor.</p>
See also	LOAD NET, CREATE NETS, FULL DRC

EXPORT SYMBOL

Purpose	To export a symbol from a file
Mode	All
Description	This command allows you to export an existing symbol from a drawing into a symbol library. It can be used to re-generate a library with symbols which have been erased or otherwise updated. Select {Library}{Export symbol} and the option menu will present the names of the symbols in current drawing. Choose the symbol you want to export, press [Return] and you will be prompted for the name of the library you want to export it to - enter the name, if different from the defaulted name and the symbol will be exported.
Keys	[Ctrl][L] {Export symbol}
Menu	{Library}{Export symbol}

FLOOD FILL

Purpose	To fill an area with copper
Mode	Block
Description	This facility allows you to define and fill an area with copper automatically avoiding any tracks and pads which have already been placed whilst paying due attention to the current global design rules. It has been implemented as an option in block mode and the fill attributes can be configured as required. Select {Config}{Block options}{Block fill setup} and you will be presented with the setup menu which has the following options :

LAYER TO FILL

This selects the layer you want to use. To change the layer, move the cursor over this option and press [Return] and the layer number will cycle through the range 1 to 8.

TRACK WIDTH

The block is filled using tracks and this option allows you to select the width of the track to be used. According to the combination of fill grid and track width, you can arrange for the block to be completely filled or hatch filled. This option shows the track width number (see {Config}{Track and pad sizes}) along with the current width setting in thou. For example : **Track width 4 (52) thou**

indicates that track width 4, which is currently set to 52 thou, will be used for filling. To change the size, move the cursor over this option and press [Return] - the track width number and associated track width will cycle through the eight available sizes.

FILL TYPE

This option determines the way in which the tracks are placed when filling. You have the choice of horizontal and vertical

hatching {HORIZ/VERT}, diagonal hatching {DIAGONAL} or a combination of both {BOTH}. If you select {BOTH} you will produce almost twice the number of tracks as either of the other options.

FILL GRID

By changing this setting you can change the pitch of the fill. The grids available are 1/10th, 1/20th, 1/30th and 1/40th of an inch - selecting 1/40th produces a denser hatching than 1/10th.

Keys

[Ctrl][O] [F]

Menu

{Option}{Flood fill}

See also

DESIGN RULE CHECK, DESIGN RULES, BLOCK

Example

To create a solid filled block on layer 1 :

Layer to fill 1

Track width 4 (52) thou

Fill type HORIZ/VERT

Fill grid 1/20th inch

To create a hatch filled block on layer 8 :

Layer to fill 8

Track width 2 (10) thou

Fill type HORIZ/VERT

Fill grid 1/20th inch

If you plan to penplot the layout it is suggested you use a hatched rather than a solid fill - it will plot faster and produce a plot with less pen overwriting.

FILLING THE BLOCK

Once the block fill settings have been made you can then proceed with the actual filling of the block.

Put a block around the area you want fill by selecting {Tools}{Block} or by pressing [F10] and dragging the block box to cover the required area - press [Return] to complete the box. To fill the box, you can either select {Option}{Flood fill} or use the direct keystroke [F]. From the menu, you will be prompted :

> Position cursor and press [Return]

Move the cursor to a vacant point within the block and press [Return] - this will be used as the starting or seed point for filling. Ideally, choose the largest space in the block which does not contain a track or pad.

To use the direct keystroke, simply position the cursor as described above and press [F].

Boardmaker will then create a filled area within the block which is connected to the seed point. If there are vacant areas in the block which cannot be connected because there are no routes to them, they will not be filled. i.e. Boardmaker will not create isolated or floating areas of copper. However, if you want to explicitly fill these areas, move the cursor to them and again press [F].

Errors

A number of error and warnings are associated with this function.

> Process may take some time. Please wait

This indicates that the fill process is working.

> Selected point is outside the block

You selected a point that was not within the defined block. What did you think the block was for ?

> Point cannot be used for seed purposes

You selected a point in the block which was not empty or was too small for the fill parameters set in the {Block fill setup} menu. Choose another point or change the setup.

> Not enough memory to complete

The combination of items already on the layout and the tracks which would be used for the hatching exceed the capacity of Boardmaker. Try reducing the hatch grid or reduce the size of the area you want to hatch.

> Hatch area is too complicated to map

You are trying to hatch a large, densely populated area of the layout using a small grid. This message should not normally occur since it is probable that the area chosen could not be sensibly filled by hand.

FORCE LAYER

Purpose To force or set the layer of a symbol reference

Mode Ref

Requirements You must be in the PCB library editor

Description When designing a symbol, you have the facility to add references to the component or symbol pins which effectively label them with certain attributes such as their name, alias etc. However, when designing a symbol which uses pads which are only on one layer, typically a surface mount device, you need to indicate this in order that the design rule checker does not produce results which are confusing. Also, if you are designing an edge connector symbol, this feature allows you to distinguish between the two pads on opposite sides of the board. Each one has a different force layer. Although this is an optional reference attribute, using it will give you more information when you are using the design rule checker.

To enter the force layer, you must first select the reference box by editing it {Edit}{Reference} and then select {Option}{Force layer}. You are then prompted :

> Please enter the layer number

Enter the layer number in the range 1-8 and this will displayed in the current reference box. *Enter 'A' for all layers if you wish to remove the force layer.*

For a full description of references, see ADD REFERENCE

Keys [L]

Menu {Option}{Force layer}

See also EDIT REFERENCE, ADD REFERENCE, NAME, ALIAS, CONNECT MODE

FULL DESIGN RULE CHECK

Purpose	To verify and apply design rules for an entire layout
Mode	All
Description	<p>This command allows you to automatically perform the final validation of your layout. Even though you may have been validating each net as you routed it, this is a 'catch all' for any changes you may have made and did not verify. In a same way as the design rule checker, it uses the net based design rules to check the clearances between item on the board.</p> <p>Select {Net}{Full DRC} to start the checking the layout and each net will be highlighted and checked. You will typically see the message :</p> <pre>> Checking net N00001 from node 1</pre> <p>This indicates that net N00001 is being checked. If a net violates the designh rules then the message :</p> <pre>> Design rule violations found</pre> <p>and the offending net will be shown. The DRC will then continue on to check the next net. However, you can interrupt it by pressing [Esc].</p> <p>If there are any violations, you can view them using NEXT and PREVIOUS VIOLATION.</p>
Keys	[Ctrl][N]{Full DRC}
Menu	{Net}{Full DRC}
See also	DESIGN RULE CHECK, DESIGN RULES Global, DESIGN RULES Net based, NEXT VIOLATION, PREVIOUS VIOLATION
Warnings	<pre>> Some net(s) not completely routed</pre> <p>Nets were found which you have not fully routed</p>

```
> Never connect
```

You connected a pin which was defined as NEVER CONNECT

```
> Always connect
```

You failed to make a connection to a pin which was defined as ALWAYS CONNECT

```
> Extra node
```

There is an extra node on the net

```
> Extra net
```

Extra nets were found on the layout

```
> Text on net! Remove before proceeding
```

This happens if there is text on a copper layer which touches a net.

GOTO ORIGIN

Purpose	To position the cursor at the floating origin.
Mode	All
Description	<p>BOARDMAKER provides two cursor modes, one of which is absolute, the other is relative. If the cursor mode is set to absolute, then the cursor readouts on the status banner are the X and Y distances from the bottom left hand corner of the drawing area - the fixed origin. If the cursor mode is relative, the cursor readouts are the X and Y distances from a floating origin set by the user. The status banner always displays the current cursor mode as an 'A' for absolute or an 'R' for relative.</p> <p>This command provides a method of moving the cursor to the floating origin. The floating origin defaults to the fixed origin in the bottom left hand corner of the drawing area.</p>
Keys	[Shift][O]
Menu	{Tools}{Goto origin}
See also	CURSOR MODE, SET ORIGIN, UNITS
Example	<ul style="list-style-type: none"><input type="checkbox"/> Select GOTO ORIGIN by pressing [Shift][O] and the cursor will move to the bottom left hand corner.<input type="checkbox"/> Now move the cursor back to the middle of the drawing area and set the floating origin by pressing [O]. The origin is now set to the current cursor position but the cursor readout on the status banner is still in absolute mode, indicated by the 'A'.<input type="checkbox"/> Change the CURSOR MODE to relative by pressing [Shift][M] and provided the cursor has not moved, the cursor readout should now read 0,0. This is now the new floating origin.<input type="checkbox"/> Move the cursor away and select GOTO ORIGIN by pressing [Shift][O] and the cursor will jump back to the floating origin you just set.

GRID

Purpose	To turn the visible grid on and off.
Mode	All
Description	<p>BOARDMAKER provides two types of component placement grid. The first is a hidden grid that the cursor snaps to as it is moved around the screen and the second is a visible grid of dots which act as a guide and reference on the screen.</p> <p>This command allows you to turn the visible grid on and off as required, although at zoom level 7, the grid will always be turned off since a 1 inch grid at this resolution is of no value. Similarly, the visible grid will change dependent on the zoom level - the lower the magnification or zoom level, the lower resolution of visible grid that is displayed.</p> <p>At zoom levels 1-4 the visible grid is 0.1 inch and for zoom levels 5-6 the visible grid is 1 inch.</p>
Keys	[Shift][G]
Menu	{Tools}{Grid off}
See also	GRID SNAP, NEAREST PAD AND NODE SNAP, ZOOM, UNZOOM
Example	<ul style="list-style-type: none"><input type="checkbox"/> Select Zoom level 3 by pressing [3]<input type="checkbox"/> Turn the grid on by pressing [Shift][G].<input type="checkbox"/> Use ZOOM [Z] and UNZOOM [U] to see the effect of the visible grid at the various zoom levels.

GRID SNAP

Purpose	To change the size of the snap grid.
Mode	All
Description	BOARDMAKER provides two types of component placement grid. The first is a hidden grid that the cursor snaps to as it is moved around the screen and the second is a visible grid of dots which act as a guide and reference on the screen. Both these grids can be turned on or off if required.

As the cursor is moved over the screen the default grid that it snaps to is 0.1" but this can be changed to suit your needs. The settings available are :

1/10th inch
1/20th inch
1/30th inch
1/40th inch
1/60th inch
1/80th inch
Free hand

The current setting of the grid snap is displayed on the status banner at the bottom the display. When the freehand selection is chosen the cursor no longer snaps to a grid and allows you place items on the board very accurately.

The cursor can also be made to move in a different way using the CURSOR TRAVEL command by holding the [Alt] key and cursor keys simultaneously. This makes the cursor glide rather than have a snap action.

Keys	[G]
Menu	{Tools}{Grid snap}

See also	GRID, NEAREST PAD AND NODE SNAP, CURSOR TRAVEL, UNITS
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Select grid snap by pressing [G] and the grid snap options will appear in the Options menu. <input type="checkbox"/> Move the menu banner over the 1/20th grid snap setting and press [Return] to select it. <input type="checkbox"/> Select a ZOOM level 2, to see what is happening more easily. <input type="checkbox"/> If the visible grid is not turned on, do so by selecting GRID - pressing [Shift][G] will toggle the visible grid on and off. <input type="checkbox"/> Select Add pad by pressing [F4]-(XT) or [F6]-(AT) and a pad will appear where the cursor was positioned. <input type="checkbox"/> As you move the cursor to a new position and press [Return], notice that the pad is placed either on the visible grid of dots or half way between them. This is because the visible grid is 1/10th and the snap grid has been set to 1/20th. In other words, the snap grid is twice the resolution of the visible grid.

HIDE/SHOW DESIGNATOR

Purpose	To hide or show the current symbol designator
Mode	Symbol
Description	<p>Sometimes it may be preferable to hide the component designators which are on either of the silk screen layers. By default, BOARDMAKER will show all designators but you may decide you want to add your own on a copper layer or indeed, do not plan to produce a silk screen for your layout.</p> <p>When you select this option, it will toggle the designator on and off.</p>
Keys	[H]
Menu	{Option}{Designator ...}{Hide/show}
See also	EDIT SYMBOL, DESIGNATOR POSITION, ANTICLOCKWISE, SIZE, WIDTH
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Load the sample board 'PLACED.PCB' into the PCB editor <input type="checkbox"/> Move the cursor near one of the symbols and press [F7]-(XT) or [F4]-(AT) to edit it <input type="checkbox"/> Select HIDE/SHOW DESIGNATOR by pressing [H] and the designator for the symbol will be hidden. <input type="checkbox"/> Press [H] again and the designator will re-appear.

HIDE NETS

Purpose	To hide the nets
Mode	All
Description	<p>This command provides a method of hiding all the nets or a selected net. The nets are displayed as thin white lines in a 'ratsnest' form with the nodes marked as diagonal white crosses. When you select {Net}{Hide nets}, the {Option} menu gives you various ways to hide the nets.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Nets on component - This hides all the nets associated with one component or symbol <input type="checkbox"/> Net by node - Allows you to hide the net connected to a particular node or component pin <input type="checkbox"/> Net by name - If you select this, the {Option} menu will present the names of all the nets on the board and you can choose one by name <input type="checkbox"/> All nets - Hides all the nets on the board
Keys	[Ctrl][N]{Hide nets}
Menu	{Net}{Hide nets}
See also	SHOW NETS, NEXT NET, BACK NET, CLEAR NETS
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Load the sample board 'PLACED.PCB' <input type="checkbox"/> Show all the nets by selecting {Net}{Show nets}{All nets}. You will see the components and the nets displayed. <input type="checkbox"/> Hide all the nets by selecting {Net}{Hide nets}{All nets} and the display will now show only the components on the board.

HOME Net node

Purpose	Moves the cursor to the first node on the net
Mode	Net
Description	As nodes are placed to create a net they are numbered sequentially from the start to the end of the net. When editing a net, HOME causes the cursor to move the first node placed (panning if necessary).
Keys	[H]
Menu	{Option}{Home}
See also	EDIT NET, DELETE, END, BACK, NEXT, NET NAME, OPTIMIZE, CURRENT
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Add a new symbol pressing [F8] and enter 'DIP14' at the prompt. <input type="checkbox"/> Select ADD NET [Shift][F2]-(XT) or [Shift][F5]-(AT), enter the net name as 'N00001' [Return] and click on 4 of the symbol pins to make up a net of four nodes. <input type="checkbox"/> Move to the home node by pressing [H]. <input type="checkbox"/> The cursor will jump to the first node you added.

HOME Track node

Purpose	Moves the cursor to the first node of the track.
Mode	Track
Description	When a track is placed in position, it is made up of a series of track segments with a node at either end. These nodes are numbered sequentially from the start to the end of the track. When editing a track, HOME causes the cursor to move the first node placed (panning if necessary).
Keys	[H]
Menu	{Option}{Home}
See also	END, BACK, NEXT, INVERT ORDER, DELETE
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Select Add track by pressing [F2]-(XT) or [F5]-(AT). <input type="checkbox"/> Move the cursor to create a new track segment and press [Return] to fix it in position. <input type="checkbox"/> Repeat this several times to create a number of track segments and note that the information banner shows the number of nodes you have created. <input type="checkbox"/> Press [H] and observe the cursor moves to the first node you placed on this track i.e. the home node.

INCREMENT Alias

Purpose	To increment the reference alias
Mode	Ref
Requirements	You must be in the PCB library editor
Description	<p>This command allows you to increment characters and numbers in the reference alias. When designing a symbol, you can use the REPEAT command to quickly copy references you have already placed to a new position complete with name and alias. INCREMENT ALIAS allows you to easily alter the alias if it is similar. For example, if it is currently 'ADDR6' it will increment to 'ADDR7', and so on. To do this, you must first select the reference, if it not already selected, by editing it {Edit}{Reference} and then selecting {Option}{Increment alias}, will increment it. This greatly speeds up the process of creating symbols which require references.</p> <p>If you try to increment when the last character is a 'Z' or 'z' you will get the error Increment/decrement failure.</p> <p>For a full description of references, see ADD REFERENCE</p>
Keys	[Shift][I]
Menu	{Option}{Increment alias}
See also	EDIT REFERENCE, INCREMENT NAME, DECREMENT NAME

INCREMENT Name

Purpose	To increment the reference name
Mode	Ref
Requirements	You must be in the PCB library editor
Description	<p>This command allows you to increment characters and numbers in the reference name. When designing a symbol, you can use the REPEAT command to quickly copy references you have already placed to a new position complete with name and alias. INCREMENT NAME allows you to easily alter the name if is similar. For example, if a pin number on an IC is 9, it will increment it to 10, and so on. To increment it, you must first select the reference, if it not already selected, by editing it {Edit}{Reference} and then selecting {Option}{Increment name}, the name will increment it. This greatly speeds up the process of creating symbols with references.</p> <p>If you try to increment when the last character is a 'Z' or 'z' you will get the error Increment/decrement failure.</p> <p>For a full description of references, see ADD REFERENCE</p>
Keys	[I]
Menu	{Option}{Increment}
See also	EDIT REFERENCE, INCREMENT ALIAS, DECREMENT NAME

INCREMENT Text

Purpose	To increment the current text
Mode	All
Description	<p>This command allows you to increment characters and numbers in text. When placing numbers and letters, for example on connectors or IC pin numbers, it allows you to easily increment the last character of the text string - if it is a number the whole number will increment. e.g. IC99 becomes IC100, IC2a becomes IC2b, Conn-R becomes Conn-S.</p> <p>If you try to increment when the last character is a 'Z' or 'z' you will get the error Increment/decrement failure.</p>
Keys	[I]
Menu	{Option}{Increment}
See also	DECREMENT, SIZE, WIDTH, KILL
Example	<ul style="list-style-type: none"> <input type="checkbox"/> Select Add text by pressing [F6]-(XT) or [F7]-(AT) <input type="checkbox"/> Enter some text e.g.'IC99' [Return] <input type="checkbox"/> Move the cursor to the right and REPEAT it by pressing [R]. You will now see IC99 displayed twice. <input type="checkbox"/> Increment the text by pressing [I] and you will now see the current text displayed as IC100

INSERT/MOVE NODE

Purpose	To move an existing track node and its associated segment(s).
Mode	Track
Description	<p>BOARDMAKER provides two methods for editing tracks. You can either INSERT nodes at the current node or MOVE the current node to a new position. These two node/track editing modes provide you with a great deal of flexibility. The edit mode is shown on the information banner at the bottom of the screen as either an M for MOVE or an I for INSERT and although these modes are selected using the menus, the quick way to toggle between them is by using the [Ins] key.</p> <p>When a track is placed in position, it is made up of a series of track segments with a node at either end of the segment. These nodes are numbered sequentially from the start to the end of the track. When editing an existing track, this command provides a method of moving the current node to a new position and dragging the connected track segment(s) with it while still maintaining connectivity to the other track segments. The operation of this command is clearer when the rubberbanding function is enabled since you can see the new position of the tracks.</p> <p>If you are moving a node in the middle of a track you will be re-routing the two track segments joined to it. If you select the END or HOME node, you will, by definition only drag only one track segment to another place on the board.</p> <p>If you are inserting a node, track segments will be inserted between the current node (n) and what was originally the next node (n + 1). The information banner will keep you up to date with the total number of nodes in the track and the current node number.</p>

Keys	[M]
	[Ins] to toggle INSERT/MOVE mode
See also	HOME, END, BACK, NEXT

Example

- ☐ Select Add track by pressing [F2]-(XT) or [F5]-(AT) and place a number of track segments by moving the cursor and pressing [Return].
- ☐ Move the cursor back along the track to one of the intermediate nodes using the BACK [B] command.
- ☐ Select MOVE NODE by pressing [M] or the [Ins] key - the information banner will display an 'M' if move mode is selected.
- ☐ Now move the cursor and notice how two track segment guide lines joined to the current node move with the cursor.
- ☐ Press [Return] to place the new track segment positions.
- ☐ Now select Insert mode by pressing the [Ins] key and again move the cursor. Notice this time that as you press [Return], new nodes and track segments are placed.

Appendix A Display driver support

BOARDMAKER has been designed to take advantage of various video display cards and does not normally require any user intervention to select the relevant driver - the display adaptor is automatically detected. This allows the user to run the software on a number of machines without having to perform an installation procedure for each.

Colour Graphics Adaptor (CGA)

320 x 200 four colour Mode 5

Multi-colour Graphics Adaptor (MCGA)

320 x 200 6 colour Mode 19

Hercules Graphics Adaptor (HGA) Monochrome

Emulated 320 x 200 4 gray scales (like CGA)

Enhanced Graphics Adaptor (EGA)

640 x 350 6 primary user colours Mode 16

Video Graphics Array (VGA)

640 x 480 6 primary user colours Mode 18

To over-ride the auto display driver detection you need to enter the following the start BOARDMAKER :

- ☐ For CGA type 'BM -ADAPC'
- ☐ For MCGA type 'BM -ADAPM'
- ☐ For HGA type 'BM -ADAPH'
- ☐ For EGA type 'BM -ADAPE'
- ☐ For VGA type 'BM -ADAPV'

Appendix B - Printer and plotter support

Laser printers

BOARDMAKER can output artwork onto Hewlett-Packard Laserjet II or compatible printers with a resolution of 300 dots per inch. In order to produce a print a large drawing at this resolution, you must be sure you have enough internal printer memory. Also, laser printers are notoriously poor when it comes to absolute accuracy (typically +/- 1.5%), but have good repeatability. BOARDMAKER provides independent X and Y scaling to cope with this. To set it, first create a test pattern in the PCB editor, say a 0.1 inch grid and then print it. Measure the results and adjust the scaling factors to suite. Repeat this until the grid is accurately printed.

Printers

This section contains a list of the printers that work with BOARDMAKER. If yours is not on the list, consult your printer dealer and ask which of these it is compatible with.

Amstrad DMP 3000	Canon PW1156A	Epson FX85
Amstrad DMP 4000	Centronics Pageprinter 8	Epson FX105
Amstrad 3160	Citizen 120-D	Epson FX1000
Amstrad LQ3500	Citizen 15E	Epson LQ500
Amstrad LQ 5000	Citizen HQP45	Epson LQ850
Brother M1709	Citizen LSP10	Epson LQ1000
Brother M1509	Epson EX800	Epson LQ1050
Canon BJ130	Epson FX80	Epson LQ1500

Epson LX80	Kyocera F1200	Panasonic P1081
Epson LX86	Mannesman Tally MT85	Panasonic KX1091
Epson LX800	Microline 84	Sameco DX135
Epson MX100	NEC P7	Star LC24-10
Epson RX80	NEC P2200	Star NB24-10
Epson RX100	HP Laserjet II	Star NB24-15
Fujitsu DPL 24	IBM Proprinter II	Star NL-10
Fujitsu DPM9G	OKI 192	Star SD-15
Fujitsu DX2100	Panasonic KXP1080	Teco YP1814

Plotters

This section contains a list of the plotters that work with BOARDMAKER. If yours is not on the list, consult your plotter dealer and ask which of these it is compatible with.

Calcomp A1	Graphtec MP4300	Roland DXY885
Calcomp 103	Houston DMP42	Roland DXY880
Graphtec MP3100	Houston DMP55	Roland DXY980
Graphtec MP3200	HP 7474	Roland DXY1100
Graphtec MP3300	HP 7475A	Roland DXY1300
Graphtec MP4100	HP7550	
Graphtec MP4200	HP 7580	

Appendix C - Error messages

> Already part of this net.

You are trying to add the same node to the current net

> Attempting to clear space. Please wait.

You probably don't have enough space on your disc. BOARDMAKER will attempt to create space by removing backup files *.PC1 or *.SCH

> Block selected exceeds paper width.

You are trying to output a block which exceeds the printer width set

> Cannot find symbol

You are attempting to erase a symbol from a library which isn't there

> Cannot repeat at the same position.

BOARDMAKER doesn't allow you to place items directly on top of each other. If you do need to do this, edit the item, repeat it in a new position, and move it back to the original position.

> Component has no reference fields.

You are trying to add a component pin to a net, but the component has no reference fields.

> Conversion failed.

You tried to load a symbol into the library editor and there were already items loaded - the result would have had too many different track and pad sizes.

> Drawing area is empty.

You have selected an output option and there is no drawing loaded.

> **Drive and/or disk problem. Abandoned.**

BOARDMAKER has encountered a problem with reading or writing on your disk.

> **File could not be found.**

You are attempting to load a file which isn't there.

> **File could not be opened.**

The file you are loading is potentially corrupt.

> **File name is too long.**

You entered an illegal DOS filename or path.

> **Filename contains illegal character.**

Illegal file name characters are SPACE, | * ? " : see your DOS manual for details.

> **Format either eg. IC9,25 or eg. TR6**

You have entered a designator name, pin name in the wrong format.

> **Hatch area is too complicated to map.**

You are trying to hatch a large, densely populated area of the layout using a small grid. This message should not normally occur since it is probable that the area chosen could not be sensibly filled by hand.

> **Highlighted items on more than one net!**

The highlighted structure shorts two or more nets together.

> **Highlighter failure.**

This is an unexpected highlighter error - contact Tsien.

> **Increment/decrement failure.**

You are trying to increment Z,z or a very big number! Or you are trying to decrement A,a or 0.

> **Library has no entries.**

There are no symbols in this library.

> **Library is full.**

Libraries can contain up to 250 equivalent IC's - try rationalizing your libraries or creating some new libraries.

> **Memory is full. Check CONFIG screen.**

You have reached the limits of BOARDMAKER.

> **Multiple net. Please try another point.**

You have selected a point to start the highlighter which

> **Must never connect this pin.**

You are adding a net node to a pin which has a connect mode set to NEVER

> **Net has less than two nodes - removing.**

You have created a net with two or less nodes. This is invalid so it is removed.

> **Net name must be unique.**

You have entered a net name which already exists.

> **No designator found within range.**

You have tried to edit a designator and none could be found within range.

> **No match found.**

You have pressed [Return] on an empty line expecting to see a list of the available filenames or symbol names in the {Option} menu. BOARDMAKER could not find any such files.

> **No net node found within range.**

You have tried to edit a net and none could be found within range.

> **No pad can be found within range.**

You have tried to edit a pad and none could be found within range.

> **No reference point found within range.**

You have tried to edit a reference and none could be found within range.

> **No symbol can be found within range.**

You have tried to edit a symbol and none could be found within range.

> **No text can be found within range.**

You have tried to edit a text and none could be found within range.

> **No track can be found within range.**

You have tried to edit a track and none could be found within range.

> **Not available from graphics screens.**

You cannot use [Print screen] from graphics screens.

> **Not available in library editor.**

The selected option is not valid in the library editor.

> **Not enough memory to complete function.**

The resultant operation would exceed the memory limits of BOARDMAKER.

> **Not enough nodes for operation.**

You are attempting to repeat a track with less than two nodes.

> **Nothing to highlight at this point.**

You selected net highlight but there were no items at the cursor position.

> **Nothing to snap to here.**

Nearest pad and node snap could not find anything to snap to.

> **Only available within library editor.**

Certain operations are only available in the library editor. e.g. Save, Erase symbol, Load etc

> **Only one designator allowed.**

When you are designing a symbol, it can only have one designator.

> **Operation will move items outside border**

You are attempting to move or copy an item which would result in part of it exceeding the drawing area.

> **Option is not appropriate.**

You selected a function which is not appropriate for schematic drawing - NC drill, Photoplot etc.

> **Please centralize your drawing.**

You have selected an output option and part of an item is overlapping the available drawing area.

> **Point cannot be used for seed purposes.**

When choosing a seed point for flood fill, it must be unoccupied. Try choosing another point.

> **Point selected is beyond border.**

You were asked to Position cursor and press [Return] and a point was selected which was outside the drawing area.

> **Problem defaulting designator.**

You have added or repeated a symbol and the designator could not be automatically incremented e.g. it is longer than 16 characters long.

> **Problem loading symbol.**

Whilst browsing, BOARDMAKER requires an amount of memory as a screen buffer. If you have a very large drawing, the resultant memory requirement may exceed that which is available - you will then get this message.

> **Problem with printer or print abandoned.**

You pressed [Esc] to interrupt a printout or the printer could not be found.

> **Process interrupted and abandoned.**

You pressed [Esc] to interrupt a net highlight or design rule check.

> **Reference must have a (unique) name.**

When designing a symbol with references, they must have unique names - rename the duplicate one.

> **Reference must point to a (unique) net.**

The reference must point to a pin and it must be the only one pointing to that pin.

> **Reference names of new symbol must match**

When updating symbols, the reference pin names of the new version must match those of the symbol you are updating.

> **Resulting track would go outside border.**

You are attempting to repeat or shift a track in such a way that part of it would fall outside the available drawing area.

> **Selected layer is not turned on.**

You have tried to place an item on a layer which is disabled. Turn the layer on by selecting {Config}{Layer selection}.

> **Selected point is outside block.**

You choose a seed point for the flood fill option which was outside the block.

> **Some net(s) not completely routed.**

This is a Full DRC message indicating that there are some nets or sub-nets which have not been routed

> **Specify centre and one point for circle.**

You are endeavouring to create a circle from a track that doesn't have exactly two nodes.

> **Symbol designators must be unique.**

You have given a component a designator which already exists somewhere on your layout.

> **Symbol has no designator.**

You are trying to change a component designator for a symbol with no designator.

> **Symbol not found.**

You have asked for a symbol from a library and it wasn't there.

> **Symbol table is full.**

You have tried to add one to many symbols to your drawing.

> **Symbol would have a part outside border.**

If you placed the chosen symbol, part of it would fall outside the available drawing area.

> **Symbols with references need designators**

You have designed a symbol with references but have not given it a designator prefix.

> **Text on net! Remove before proceeding.**

There is text on the highlighted net. BOARDMAKER cannot perform a design rule check until it is removed.

> **The points specified must be different.**

When creating circles, you must specify exactly two track nodes which are not at the same point.

> **The track cannot be split at this node.**

You are attempting to split the end of a track - this makes no sense.

> **There are no nets.**

You have requested a net operation and the layout contains no nets.

> **This designator/reference item is full.**

You should not normally see this message - if you do contact Tsien

> **This is not a valid BOARDMAKER file.**

You attempted to load a file which was not a BOARDMAKER PCB or schematic

> **This node is already part of a net.**

Whilst adding or editing a net, you have selected a node which is already on it.

> **Through pad conflicts with another.**

You are trying to place a pad which is on all layers directly over another.

> **Too many segments in this track.**

The track you are creating has more than 250 segments.

> **Total clearance exceeds maximum.**

You have got a large track with excessive design rules.

> **Turn rubberbanding off in Block options.**

You are trying to copy or kill a block and the {Capture mode} is set to rubberbanding on.

> **Unexpected condition-call Tsien (UK) Ltd**

General fatal error message and something strange has happened - call Tsien

> **Unknown reference field. Please remove.**

Complete reference failure !!!!! - call Tsien.

> **Unrecognised designator.**

You are adding a designator by name and it could not be found.

> **Unrecognised pin name.**

You are adding a designator by name and the pin name entered could not be found.

> **Valid layers are 0 or 9.**

Component designator can only be placed on layers 0 or 9.

> **Warning! Artwork could not be saved.**

There is no room to save the drawing on your disk.

> **Warning! Extra nodes on net.**

The highlighted structure generally agrees with the net, but there are extra nodes which are not part of any other net.

Index

A

Absolute cursor	56
Add	
designator	97
net	64, 99
pad	31, 102
reference	103
symbol	35, 105
text	32, 106
to library	108
track	27, 107
Alias	110
definition	15
All layers	30, 38, 111
All width	38, 113
Angle assist	20, 30, 114
Anticlockwise	
block	51, 120
designator	115
designator prefix	116
pad	119
symbol	118
text	41, 117
Arc	36 - 37, 122
Assist flip	31, 124
Auto pan	73, 125
Auto via	29, 126

B

Back	
net node	128
track node	129
Back a node	38
Back modify	25
Back net	127
Backup timer	75, 130

Bill of materials	25
Block	
anticlockwise	120
capture mode	48
clockwise	138
defining	47
delete	49
drawing	47
fill	52, 181
kill	205
leave items	48
maintaining connectivity	49
manipulation	47
mirror	51, 224
mode	131
move	49
origin	48 - 49, 133
penplot	51, 250
pickup items	48
print	51, 257
repeat	50, 263
rotate	50
rotate anticlockwise	51
rotate clockwise	50
setting block options	48
shift	281
Block options	48, 51, 132
Browse libraries	76, 135

C

CAM	25
Capture mode	48
CGA/HGA display	12
Change orders	23
Circle	36, 122
Clear nets	67, 136
Clear violations	137
Clearing the workspace	75
Clockwise	
block	50, 138

Colours	55	pad	40
Component placement	22	symbol	47
Component values	44	text	46
Configuration	54	track	45 - 46, 158
changing	82	track segment	45
saving	84	Design rule check	69, 162, 186
Connect mode	140	Design rules	
Connectivity	49	definition	17
Conventions	7	global	160
Country	203	net based	161
Create nets	25, 67, 141	real-time display	261
Creating ground planes	52	Designator	
Current	17	definition	15
designator prefix	142	add	43, 97
Net	143	anticlockwise	115
pad	144	edit	43, 167
reference	145	hide/show	44
symbol	146	mirror prefix	226
text	147	moving	43
track	148	position	159
Current settings	54	prefix size	286
Cursor		prefix width	305
box	149	rotating	43
mode	20, 56, 150	size	44, 285
settings	56	text content	293
size	56, 151	width	44, 304
travel	152	Designator prefix	
type	153	anticlockwise	116
X & Y locking	230	current	142
		kill	206
		layer	214
		text content	292
		Designator:hide/show	192
		Directory	75, 164
		setup	75, 165
		working	75
		Dongle	
		Protection device	6 - 7
		Dot-matrix printer	84
		settings	85
		DRC	17, 162, 186

D

Decrement	
text	156
Decrement alias	154
Decrement name	155
Decrement text	34
Delete	
block	49
net node	66
node	157

E

Edit	
designator	167
net	65, 168
pad	40, 170
reference	171
symbol	42, 172
text	40, 173
track	174
End	
net node	175
track node	176
End node	38
Erase symbol	177
Exit	20, 178
Export net	179
Export nets	68
Export symbol	180

F

Filling an area	52
Flood fill	52, 181
Force layer	185
Full DRC	186
Function keys	8

G

Global design rule checker	72
Global design rules	69
Global DRC	24
Goto origin	188
Grid	189
nearest node	238
nearest node snap	58
nearest pad	238
snap	58, 190

visible	57
Grid settings	57
Grid snap	20, 57 - 58
Ground plane	181
delete	60
Group layers	86

H

Hide net	69
Hide nets	193
Hide/show designator	192
Home	
net node	194
track	195
Home node	38

I

Increment alias	196
Increment name	197
Increment text	34, 198
Information banner	18
Insert node	199
Installation	6
Invert order	46, 201
Inverted output	86

K

Keyboard	8
Keyboard country	203
Keyboard layout	204
Keyboard settings	84
Kill 46	
block	205
designator prefix	206
highlighted	207
highlighted net	60
pad	209

reference	210
symbol	211
text	212
track	213
Kill net	208

L

Laser printer	86
Layer	31, 33
all	30
changing	29, 38
pad	31, 40, 215
pairs	216
segment	29, 276
text	217
Layer colours	55
Layer grouping	86
Layer pairs	29, 56 - 57, 216
Layer selection	20, 30, 55, 218
Layers	
changing all	111
Layout editor	20
Library	
adding to	108
browse	76, 135
creating a new library	81
creating a new symbol	77
editing a symbol	82
erasing a symbol	81
updating a symbol	82
using	76
viewing	77, 302
Library editor	20, 76
capacity	76
Line editor	10
Load	219 - 220
file	74
Load nets	62

M

Macros	35
Main menu	9, 20
Managing PCB files	74
Manual checking	25
Manufacture	25
Menu banner	13
Message timer	83, 223
Messages	10
Mirror	
block	51, 224
designator prefix	226
pad	227
symbol	44, 228
text	41, 229
Mode	
block	19
current	17, 149
cursor	150
pad	19
symbol	19
text	19
track	18
Mouse	
sensitivity	83, 231
type	83, 231
Mouse locking	83, 230
Mouse settings	231
Move	
block	49
node	199
symbol	42

N

NC drill	90, 234
Nearest node	58
reference	237
Negative output	86

Net		definition	15
add	99	New	75, 239
back	127	Next	
back a node	65, 128	net node	241
clear	67	track node	242
clearing	136	Next net	240
creating net list	141	Next node	38
creation	21	Next violation	243
current	143	No mode selected	12
definition	15	Node	
delete	66	delete	157
design rules	66		
edit	168		
editor	63		
end node	65, 175		
export	68, 179		
hide	69		
hiding	193		
home node	65, 194		
kill	208		
loading	62		
name	66, 232		
next	240		
next node	65, 241		
optimize	69, 245		
rip-up	207		
show	69		
showing	284		
view	67		
viewing a node	301		

Net based design rules	66
Net highlight	235
Net highlighting	58
warnings	59
Net list	60
Net list editor	61, 63
Net list format	221
Net mode	19
Net name	232
Net rip-up	60
Netlist	

O

Optimize net	69
Optimize nets	245
Option menu	12
OrCad	220
Origin	
block	48, 133
fixed	56
floating	56
goto	188
setting	56, 279
Output	
dot-matrix printer	84
NC drill	234
penplot	247, 251
print	255
Output mode	85
Output scale	85

P

Pad	
add	102
anticlockwise	119
current	144
delete	46
edit	170
fixing	32

Index

kill	209	add	103
layer	215	alias	110
mirror	227	box	264
moving	40	current	145
repeat	53, 267	decrement alias	154
shape	31, 280	decrement name	155
width	32, 306	edit	171
Pan	246	force layer	185
auto pan	125	increment alias	196
using the mouse	73	increment name	197
Parallel port	7	kill	210
Penplot	247	name	233
Penplot block	51, 250	nearest node	237
Penplotter	86	repeat	265
settings	87	size	287
Photoplot	89, 251	References	
Plot setup	88	connect mode	140
Position designator	159	Relative cursor	56
Previous violation	254	Repeat	
Print	255	block	50, 263
Print block	51, 257	pad	53, 267
Print style	86	reference	265
Printer		symbol	54, 268
mode	85	text	54, 269
scale	85	track	53, 266
settings	85	Report	24
type	85	Reports	25, 270
width	85	Rotate	
Printer settings	258	block	120
Printing library contents	77	pad	119
Problems	7	symbol	118
Prompts	10	text	117
		block	50, 138
		designator prefix	116
		symbol	44
		Routing tracks	23
		Rubberband	271

Q

Quasi-track 261

R

READ.ME file 6
Reference 79

S

Save	272
file	74
Save options	274
Schematic capture	21
Schematic editor	20
Segment layer	38, 276
Segment width	28, 38, 277
Selecting tracks	18, 38
Selection banner	9
Set origin	56, 279
Settings	54
Shape	31, 280
pad	40
Shift	
block	281
track	283
Show net	69
Show nets	284
Size	33
cursor	151
designator	285
text	288
Sizes	
track and pad	294
Snap grid	190
Solder resist	
clearance	86
Specification	4
Split track	38, 46, 289
Starting BOARDMAKER	7
Status	73
Status banner	19
Structure rip-up	207
Surface mount	51
Symbol	
add	105
anticlockwise	118
browsing	135

creating	77
current	146
delete	47
edit	42, 172
erase	177
exporting	180
fixing	36
kill	211
mirror	44, 228
moving	42
repeat	54, 268
rotate	44
update	298
value	44

T

Tags	73, 290
Tango	220
Text	
add	106
anticlockwise	117
changing	42
content	42, 291
current	147
decrement	156
delete	46
designator	293
edit	173
fixing	34
increment	198
increment/decrement	34
kill	212
layer	33, 41, 217
mirror	41, 229
moving	42
repeat	54, 269
rotating	41
selecting	41
size	33, 41, 288
width	33, 41, 303

Timer	
backup	75, 130
message	83, 223
Track	
add	107
current	148
delete	45, 158
edit	174
home node	195
insert	39
kill	213
moving segments	39
moving/shifting	38
quasi-mode	261
repeat	53, 266
shift	283
split	38, 289
width	28, 38
Track/ pad sizes	28, 32, 54, 294

U

Units	57, 296
Unzoom	72, 297
Update symbols	82, 298
Usage window	74
Using symbols	34

V

Value	44
Verifying routes	162

Via hole	
auto placement	56
pad size	55
View library	302
View node	301
Viewing nets	67
Violation	
clear	137
next	243
previous	254

W

Width	32 - 33
all	28
designator	304
designator prefix	305
pad	40, 306
segment	28, 277
text	303
track	113

X

X-bias	69
--------	----

Z

Zoom	20, 72, 307
------	-------------

BOARDMAKER Guide to Release 2.30

This document describes the enhancements added up to and including release 2.30. It contains a general overview and description of each function enhancement in detail. Insert this section in the front of your manual and file the command pages alphabetically in the Command Summary.

Enhancement Summary

- Additional netlist translators
- Automatic component mapping
- Auto via removal
- Block copy allows for designator REPEAT or INCREMENT
- Component find
- Component renumber and backannotation
- User definable default net attributes
- Design rule modes
- DRC error flags
- DRC ignore layers
- Edit net attributes
- Extended reports generator
- Imperial or metric track and pad sizes
- Multiple backup copies
- Pad matrix facility
- PostScript output driver
- Symbol rename
- Track capacity optimiser (compressor)
- Top down modification

Additional netlist translators

The following netlist formats are now supported :

- ☐ Tango (as produced by OrCAD version 3.0 or higher)
- ☐ Mentor

- ☐ Racal Redac (includes Cadnetix and OrCAD version 1.0 or ...gher)
- ☐ Protel (as produced by Protel Schematic and Tango Schematic)
- ☐ Schema (as produced by Schema II and III)

Further formats are being developed. Please contact our support department for details.

Installation notes

If you already have a previous version of BoardMaker, version V2.23 and BoardRouter must be installed in your existing BoardMaker directory. The installation procedure defined below, involves copying all the new executable files to your working directory. None of your configuration, library or artwork files will be affected.

To install V2.23 and the BoardRouter software, please complete the following steps (typical examples are shown):

1. Change directory to your BoardMaker working directory.
`cd \bm2 [Return]`
2. Put System Update Disk 1 in drive a: or b: and log onto that drive.
`a: [Return]`
3. Type INSTALL source_drive destination_drive [Return].
`install a: c: [Return]`

Now follow the on screen instructions to install the software.

If you are updating from BoardMaker to BoardRouter, you now need to telephone Tsien Technical Support on (0)223 277767 for details of how to remotely enable the autoroute software. You will need to be next to your computer and please quote your invoice number.

Autorouter software

BoardRouter is a fully re-entrant, gridless autorouter which integrates directly with BoardMaker 2 and it is accessed from the {Net} menu. The {BoardRouter} option will only appear in this menu if the correctly programmed security key and autoroute software have been installed.

BoardRouter is designed to allow you to effortlessly switch between manual routing and autorouting. When the nets have been loaded and the

components placed using BoardMaker, you can manually edit items as required. When you select {Net}{BoardRouter}, BoardMaker will carry out a full design rule check to ensure nothing is in error and then pass control to BoardRouter.

Various setup options are available, including :

- ☐ Layer routing direction (HORIZONTAL, VERTICAL or NOT USED).
- ☐ Nets to be routed can be turned on or off.
- ☐ Automatic file save options.
- ☐ Route by net or the whole board.

Because BoardRouter is gridless, the minimum of routing setup is required, since the tracks and vias are routed and placed based on the information contained with each net. ie track width, design rules, via size and net type.

Internally, BoardRouter sequences the order nets are routed by costing the length, width (including design rules) and type of net. These costings change dynamically, but are initially governed by the net type. Nets are nominally routed in the following order :

- ☐ BUS
- ☐ POWER
- ☐ SIGNAL
- ☐ UNSPECIFIED

These can be changed either in the net list or by editing the net in BoardMaker to change the net type giving you high-level control over the order tracks are routed in.

Whilst routing, the status banner displays 'Working' and can be interrupted by pressing [Esc]. It is then possible save the routes placed and exit back to BoardMaker. This switching between BoardMaker and BoardRouter can be done any number of times giving you the freedom to take advantage of both manual and autorouting.

Automatic component mapping

The mapping facility when loading nets has been vastly improved. Once a particular mapping has been done, a MAP file is created. This is an ASCII file which details the component name followed by the path name to the

BoardMaker symbol it is mapped to. This MAP file can then be used to automatically map your components on future occasions.

Auto via removal

Vias which were placed automatically are now effectively part of the track they are on. Consequently, as track nodes are moved or deleted, associated vias will be moved or deleted.

Repeat block allows for REPEAT or INCREMENT

This allows you to determine whether component designators are repeated or incremented whilst copying blocks. This is designed to cater for step and repeat operations prior to manufacturing.

Component renumber and backannotation

The components can be renumbered from left to right, top to bottom to provide an ordered arrangement for component designators. A backannotation file is produced which details the changes made to allow you to backannotate your schematic. (e.g. the OrCAD BACKANNO.EXE facility).

Component find

This function provides a method of finding components by their designator.

User definable default net attributes

When converting netlists to BoardMaker format, you have control over the default settings for the design rules, track widths, via sizes and net type.

Design rule modes

This allows the real-time design rule information displayed in Quasi-track mode to behave in various modes. It also determines the design rule clearances used by the Flood Fill facility. The modes are :

- ☐ Global DRC for checking the board to one set of rules.
- ☐ Maximum DRC for flood filling.
- ☐ Net based DRC for use with the NEXT and BACK NET facility.

DRC error flags

Design rule violations are now displayed on the screen as a flag. BoardMaker graphically shows the point at which the violation has occurred and labels it. The error flags are maintained in all modes and can be cleared if required.

DRC ignore layers

You can choose the layers you want the design rule checker to work on. It allows you to use an intermediate layer for mechanical information and the DRC can be made to ignore that layer. Select {Config}{Design rules}{DRC Ignore layers} to choose the layers to ignore.

Edit net attributes

The attributes of each net (width, type, via size and design rules) can be edited either individually or on a batch basis using the Net Attribute Editor.

Extended reports generator

The reports generator has been extended to provide :

- ☐ Statistics.
- ☐ Full PCB Description.
- ☐ Component details.
- ☐ Bill of Materials.
- ☐ Output directly to file or printer ports.
- ☐ Page length control.
- ☐ Database communication files.

Imperial or metric track and pad sizes

The track and pad sizes are displayed in either imperial or metric. You can toggle this by pressing [U] when viewing the track and pad sizes screen.

Multiple backup copies

It is possible to determine the number of backup copies BoardMaker keeps when you save a file. Up to nine can be maintained and restored for the ultimate in confidence.

Pad matrix facility

This facility is designed to cater for non-standard component pin pitches. You can specify the repeat count and pitching in both X and Y direction. This is particularly useful for designing surface mount symbols.

PostScript output driver

In addition to the plotter and printer support provided by BoardMaker, V2.23 includes a driver compatible with the Adobe PostScript language. This may be used to produce artwork on any PostScript compatible device such as an Apple LaserWriter. Equally, when using suitable wordprocessing or desk top publishing package (eg WordPerfect, Ventura Publisher), quality PCB artwork can be incorporated directly into technical documentation.

Symbol rename

Symbols already loaded into a PCB can be re-named from within the PCB editor. This allows you to update a symbol from one with a different name.

Track capacity optimiser (compressor)

The compressor provides a means of optimising the track capacity. Tracks which have coincident start and end points are joined. This is particularly useful when you have placed a ground plane on previous versions and have used up the track capacity.

Top down modification

This allows the designer to introduce changes to existing schematic diagrams or netlists at any stage during the design process. By making changes to your schematic, the modifications are processed starting at the top level downwards - the corresponding changes are automatically made to the PCB. Unchanged sections of the PCB are left untouched, requiring only the areas affected by the change to be re-worked. This ensures the changes are only made once in one place and are therefore easier to monitor and importantly, the integrity between schematic and PCB is maintained.

AUTOROUTER

Purpose	To access BoardRouter
Mode	All
Keys	[Shift][F10]
Menu	{Net}{BoardRouter}
Description	When you access BoardRouter, your layout which was loaded in BoardMaker will be transferred automatically to the autorouter complete with tracks, pads, texts, symbols and nets. The screen layout for BoardRouter is similar in layout to BoardMaker.

The menu banner at the top shows the options :

File Route Net Layer Config V1.03

The status banner at the bottom shows :

R + 11.302, +8.138 1/20 Working Zoom3

This shows the cursor mode, XY cursor position, cursor grid and zoom level. The word 'Working' will appear when the router is making connections.

The information banner will show how the router is faring :

Routed:50/200(25%) Failed:2 Vias:15

This indicates that 50 from a total of 200 sub-nets have been successfully routed, 2 sub-nets have failed to route and 15 vias have been placed.

Keys	To interrupt the router press [Esc]. You can pan, zoom and unzoom using [P], [Z] and [U] respectively. Whilst routing the sensitivity of these keys will be reduced. The direct zoom level keys [1] - [7] can also be used.
-------------	---

File Menu

Selecting {File}{Exit} will exit BoardRouter and return you to BoardMaker. If any new tracks have been routed you are prompted :

Save new routes to PCB file? (Y/N).

Pressing [Y] will transfer the tracks and vias to the original PCB file. If you want to continue routing after you have interrupted it, simply start it again as required by selecting {Route}{Board} or {Route}{By net}.

{File}{Save} explicitly saves the tracks and vias to an intermediate ARB file.

Route Menu

BoardRouter lets you route a single net or the entire board.

Route by Net

When you select {Route}{By net} you will be presented with a window showing all the nets on the board. Click on the net you want to route and the status banner will display 'Working' while it attempts to make the connections. You will hear a 'plip' when a successful route is made. **If the net you selected was turned off in the {Net} menu, it will not be routed and an error message will be displayed.**

Routing the board

The {Route}{Board} option provides a means of routing the entire board. When selected, the status banner will display 'Working' while it attempts to route the board. You will see a white net line appear for the connection being routed, followed by a 'plip' when it is placed. You are able to Pan and Zoom whilst this is happening. If you want to interrupt the router, press [Esc].

Example

What follows is a typical sequence of events which allows you to take full advantage of Boardmaker and BoardRouter.

Preparation

Before you access BoardRouter, make sure you have completed the following tasks.

- ☐ If you are using a schematic capture package, convert the netlist into Boardmaker format using {Load nets}{Convert from ...}. Use the {Set defaults} option to set the track sizes, via sizes and design rules you want for the majority of your nets. Map the components and create a new PCB.
- ☐ If you are not using schematic capture, place the symbols you want to use in the PCB editor and use {Add net} to rats nest the circuit.
- ☐ Place the components in positions suitable for routing.
- ☐ Use {Edit net} to change the track widths, via size and design rules for any nets which you want to be different from the defaults. e.g. Vcc, Ground etc. BoardMaker routes nets in the order, BUS, POWER, SIGNAL UNSPECIFIED, so make sure you set the net type appropriately.
- ☐ Make sure the track and pad sizes are correct for the width numbers you have set above.
- ☐ Create an outline for the board using tracks placed on each layer that you plan to route on.
- ☐ Create any keep-out areas where you don't want any autorouted tracks and vias by manually placing tracks on each layer you plan to route on.
- ☐ Use {Config}{Layer selection} to select the colours for the layers you are using for routing.
- ☐ Manually route any nets necessary.
- ☐ Select {Net}{Autoroute} or press [Shift][F10] to access BoardRouter. A full design rule check (Full DRC) will be automatically carried out to ensure there are no errors prior to autorouting. If any are found you will need to rectify them before continuing.

Setup

Once the board has successfully passed the Full DRC, control is passed to BoardRouter ready for autorouting.

- ☐ Use the {Layer} menu to choose the routing layers and direction.

- ☐ Set the backup timer to a suitable value.
- ☐ Turn off any nets which you don't want to route now using the {Net} menu. Click on the net name until a cross appears, which indicates it is not to be routed.
- ☐ Select {Route}{Board} to start the autorouter. The status banner will display 'Working' and the information banner will give details of the number of successful routes, the number of failed routes and the number of vias placed.
- ☐ Remember you can Zoom [Z], Unzoom[U] and Pan [P] whilst the router is working. Press [Esc] if you want to interrupt it.

Tidying up

When the router has finished, the 'Working' message will disappear. The information banner will show the success rate. You are now faced with the problem of what to do with any remaining sub-nets which could not be routed. There are a number of approaches you can adopt, but first you will need to update the PCB file with the tracks and vias that have been added by the router. Save the routes and exit back to Boardmaker and perform a Full DRC [Shift][F10] which verifies and hides successfully routed nets.

Examine the board for obvious areas of congestion. You may have placed a component in an unsuitable position. Try moving it to a new position, modify or delete any tracks connected to it and try autorouting it again.

If this fails to produce satisfactory results, try moving other components which may further free up the congested area and try again. Eventually, you will reach a stage where the only solution is to manually route the remaining nets.

WARNINGS

Should BoardRouter experience an unexpected condition, it will allow you to save the work it has already completed by prompting you as follows :

Critical error : xx. Press [Return]



Please phone Tsien quoting xx.

Routes may now be safely saved. [Return]

Make a note of the error code xx and report it to the Tsien technical department. You can then safely save your work.

Net Menu

When you select this, a window will display a list of all the nets. Adjacent to each is a tick or a cross which shows whether the net is turned on or off. These default to on (a tick), but can be toggled by clicking on it. When you route the board, it will only route those nets which are ticked. You may want to use this :

- ☐ to route the entire board EXCEPT certain nets.
- ☐ to route specific sections of the circuit selectively.

Note that you will not be able to route {By net} a net that is turned off.

Layer Menu

It is normal practice in laying out PCBs to place tracks vertically on one layer and horizontally on the next. When you select {Layer} you are given the opportunity to choose HORIZONTAL tracks, VERTICAL tracks or NOT USED for each of the routing layers. Choosing either HORIZONTAL or VERTICAL does not constrain the router to produce exactly vertical or horizontal connections. BoardRouter will introduce 45 degree 'tacks' so that objects can be avoided without excessive use of vias.

Config Menu

To avoid losing any work whilst routing, the {Config}{Backup timer} allows you to set BoardRouter to automatically save it's work. This can be set between OFF and 30 MINS in 5 minute increments. Note that when it saves, it does so to an intermediate file, not directly to the PCB file.

File recovery

Should you experience a power failure on your computer while routing, all is not lost! You can recover the work from the time it was last saved by the backup timer. This backup file has the same name as the PCB file, but with the file extension .ARB.

If you are routing a PCB file called SAMPLE.PCB, the autorouter will create and update an intermediate save file called SAMPLE.ARB.

To recover the work done, use the DOS copy command to transfer the files eg.
`copy sample.arb sample.pcb`

BACKUP COPIES

Purpose	To set the number of backup copies
Mode	All
Description	<p>When a file is saved, BoardMaker keeps a number of backup copies. The main file extensions are .PCB and .SCH for PCB and schematic files respectively. By selecting {Backup settings}{Backup copies} you can select the number of backups. These are given the file extension .PC1 .PC2 etc up to .PC9, similarly for schematic files .SC1 etc.</p> <p>This procedure ensures that you can always go back to old versions you saved. If you make a severe error and need to use the backup, copy the .PCx or .SCx file to a .PCB or .SCH file using the DOS copy command.</p> <p>Old versions can be restored using the RESTORE BACKUP command.</p>
Keys	[Ctrl][C]{Backup settings}{Backup copies}
Menu	{Config}{Backup settings}{Backup copies}
See also	RESTORE BACKUP, SAVE, SAVE OPTIONS, CONFIG AUTO SAVE, MESSAGE TIMER, BACKUP TIMER

COMPONENT FIND

Purpose To find a specific component

Mode All

Description This command provides an easy method of finding a particular component on your layout as referenced by its designator. Select {Net}{Component find} and the {Option} menu will present a list of all the component designators on the board. Select the one required and the cursor will move to the origin of the component, panning if necessary.

Keys [Ctrl][N]{Component find}

Menu {Net}{Component find}



COMPONENT RENUMBER

Purpose	To automatically renumber the components on a PCB
Mode	All
Description	<p>Components can be renumbered to provide an ordered physical layout either to aid manufacture or simply for aesthetic reasons. A WAS/IS file is produced which can be used to backannotate your schematic.</p> <p>The components can be renumbered left to right, top to bottom by selecting {Net}{Renumber} and you will see the message :</p> <p>Start renumber top left or right (L/R)?</p> <p>Press [L] or [R] as required.</p> <p>When finished, the components will be renumbered and Boardmaker produces a WAS/IS in the net directory. The file extension for this file is .WAS and is in the following format :</p> <p>C2 C19</p> <p>IC3 IC1</p> <p>The first entry is the old designator, the second entry is the new designator. The two are separated by a space. i.e. what was C2 is now C19 etc. Unchanged designators will not be in the file.</p>
Keys	[Ctrl][N]{Renumber}
Menu	{Net}{Renumber}

COMPRESSOR

Purpose To optimise the track capacity

Mode All

Description The track compressor provides a means of optimising the track capacity. Track segments which have coincident start and end points are joined to make complete tracks. This is particularly useful when you have placed a ground plane on previous versions and have used up the track capacity.

Select {Tools}{Compressor} and you will typically see the message :

Packing track 23 of 342

The first number will be incrementing and the second decrementing. The process can be safely interrupted at any time by pressing [Esc]. This process may take several minutes for large PCBs. **When using the FLOOD FILL facility, you can opt to dynamically compress the tracks by pressing [Shift][F] rather than [F].**

Keys [Ctrl][T]{Compressor}

Menu {Tools}{Compressor}

See also FLOOD FILL

DESIGN RULES Ignore layers

Purpose	To turn off the design rule checker on named layers.
Mode	All
Description	You can choose the layers you want the design rule checker to work on. It allows you to use an intermediate layer for mechanical information and the DRC can be made to ignore that layer. Select {Config}{Design rules}{DRC Ignore layers} to choose the layers to ignore.
Warning	Tsien Health Warning !!! Misuse of this function can seriously confuse.
Keys	[Ctrl][C]{Design rules}{DRC ignore layers}
Menu	{Config}{Design rules}{DRC ignore layers}
See also	DESIGN RULE CHECK, DESIGN RULES Net based, DESIGN RULES Global

DESIGN RULES Mode

Purpose To change scope of the design rules

Mode All

Description This allows the real-time design rule information displayed in Quasi-track mode to behave in various modes. It also determines the design rule clearances used by the Flood Fill facility. Select {Config}{Design rules}{Design rule mode} and you can set the following modes :

Global DRC

Allows you to set the global design rules manually.

Maximum DRC

Sets the global design rules to the maximum of the net based design rules. This should be selected prior to doing a flood fill to ensure that no design rules are violated.

Net based DRC

With this mode selected, the Quasi-track design rule clearances will follow the net based design rules when NEXT and BACK NET are selected. You should consider using this as the default mode if you want to take best advantage of the net-based design rules.

Keys [Ctrl][C]{Design rules}{Design rule mode}

Menu {Config}{Design rules}{Design rule mode}

EDIT NET

Purpose To edit an existing net and its attributes.

Mode Net

Description This command provides a method of editing nets which have already been added to a layout. What you are actually doing is selecting a net prior to performing some other editing command such as changing its name or deleting a node. When the net is selected it becomes the CURRENT net and the information banner will display :
N00001,P (10,12,15) (2,3,16) Tot=6

N00001 - the net name.

P - net type. P = POWER, U = UNSPECIFIED, S = SIGNAL, B = BUS.

10 : track to track clearance in thou.

12 : track to pad clearance in thou.

15 : pad to pad clearance in thou.

2 : nominal track width number 2.

3 : minimum track width number 3.

16 : via number 16.

Tot = 6 : there are six nodes.

Since nets have a name, there are two ways of defining which net you want to edit - you can edit the net by selecting one of its nodes using {Edit}{Net} or by naming it using {Net}{Edit net}.

EDITING BY NAME

If you want to edit a net and you know its name but are unsure of its exact nodes, select {Net}{Edit net} and the {Option} menu will present you with the names of all the nets. When you select the one you need to edit, all other

nets will be hidden and the selected net will be displayed.
You are then able to modify the net as required.

EDITING BY NODE

If you want to edit a net and you know where one of its nodes is, simply put the cursor on the node and select EDIT NET

CHANGING THE NET ATTRIBUTES

- ☐ Nominal track width number. Select {Option}{Nominal width} or [W] and enter a track width number at the prompt.
- ☐ Minimum track width number. Select {Option}{Minimum width} or [Shift][W] and enter a track width number at the prompt.
- ☐ Via size. Select {Option}{Via size} or [S] and enter a pad width number at the prompt.
- ☐ Net type. Select {Option}{Net type} or [T] and choose POWER, BUS, SIGNAL or UNSPECIFIED from the {Option} menu.

Keys

[Shift][F1] to select net by node

[Ctrl][N]{Edit net} to select net by name

Menu

{Edit}{Net} to select net by node

{Net}{Edit net} to select net by name

See also

ADD NET, BACK, CURRENT, DELETE, END, HOME, KILL, NEXT, NET NAME, OPTIMIZE

FULL DESIGN RULE CHECK

Purpose

To verify and apply design rules for an entire layout

Mode

All

Description

This command allows you to automatically perform the final validation of your layout. Even though you may have been validating each net as you routed it, this is a 'catch all' for any changes you may have made and did not verify. In a same way as the design rule checker, it uses the net based design rules to check the clearances between item on the board.

Select {Net}{Full DRC} to start the checking the layout and each net will be highlighted and checked. You will typically see the message :

Checking net N00001 from node 1
This indicates that net N00001 is being checked. If a net violates the design rules then the message :
Design rule violations found

and the offending net will be shown. The DRC will then continue on to check the next net. However, you can interrupt it by pressing [Esc].

If there are any violations they will be displayed as DRC error flags on the screen and you can view them using NEXT and PREVIOUS VIOLATION. The flag will rooted to the point where the violation has occurred.

Keys

[Ctrl][N]{Full DRC}

Menu

{Net}{Full DRC}

See also

DESIGN RULE CHECK, DESIGN RULES Global, DESIGN RULES Net based, NEXT VIOLATION, PREVIOUS VIOLATION

Warnings

Track to track violation displays as T-T

Track to pad violation displays as T-P

Pad to Pad violation displays as P-P

Some net(s) not completely routed

Nets were found which you have not fully routed

Never connect (Error flag displays NEVER)

You connected a pin which was defined as NEVER CONNECT

Always connect (Error flag displays MUST!)

You failed to make a connection to a pin which was defined as ALWAYS CONNECT

Extra node (Error flag displays ?NODE)

There is an extra node on the net

Extra net (Error flag displays EXTRA)

Extra nets were found on the layout

Text on net! Remove before proceeding (Error flag displays TEXT!)

This happens if there is text on a copper layer which touches a net.

Short(s) found with net N002

This net shorts with a net called N002.

LOAD NET

Purpose

To load a net list and make or top modify a PCB

Mode

All

Description

This facility allows you to import a netlist from a schematic capture package or one generated manually in a text editor. If the netlist you want to load is not in BOARDMAKER format you will first need to translate it.

Converting netlist formats

Select {Convert from . .} and choose the format required. You are then prompted to enter the name of your netlist file and the name for the new file to be created in BOARDMAKER format. When the names have been entered the netlist will be translated. Supported formats are :

- ☐ Tango (as produced by OrCAD version 3.0 or higher)
- ☐ Mentor
- ☐ Racal Redac (includes Cadnetix)
- ☐ Protel (as produced by Protel Schematic and Tango Schematic)
- ☐ Schema (as produced by Schema II and III)

To create a Tango netlist using OrCAD use the following command :

NETLIST INPUT.SCH OUTPUT.NET TANGO
/O/S

Default net attributes

The BoardMaker netlist format allows you to set the design rules, track widths, net type and via size for each net. Consequently, when converting another netlist to BoardMaker format, these attributes are inserted. You can select the defaults used when converting the nets by selecting {Default settings}. You can change the following :

- ☐ Nominal track width number and size.

- ☐ Minimum track width number and size.
- ☐ Via number, diameter and hole size.
- ☐ The net type.
- ☐ The Design Rules.

These settings are changed using the [+] and [-] keys. Note that [Return] is the same as [+].

Loading a netlist

To load a BoardMaker netlist, select {Load net file} and you are prompted for the net filename.

If you cannot remember the name of the file or simply want to see what files are available, press [Return] and a directory window will present a list. Put the banner over the name you want and press [Return] to select it.

Checks are made for :

- ☐ Duplicate designator names.
- ☐ Duplicate net names.
- ☐ Unique net nodes.
- ☐ Nets with less than two nodes.
- ☐ Components in the netlist which are not in the component list.

Mapping the components

Once the net file has been successfully loaded, you need to map the component names to the PCB layout symbols you want. For example, a component from the schematic called 74LS00 could be mapped to a layout symbol called DIP14.

Select {Map symbols} and you will be presented with two windows. The top window gives details about the component in the netlist, giving its designator, symbol name and the pin names which have been used. The bottom window gives details of the PCB layout symbol you want to map it to. You can use [PgUp], [PgDn], [Home] and [End] to flip through each of the components in the net

list. [F1] allows you to scan to the next unmapped symbol and [F2] can be used to search for a particular component.

To map the symbol, press [Return] and you will be prompted for library name. Enter this directly or press [Return] to see a directory. When you have selected a library name, the window will show a list of the symbols in the library.

Put the cursor over the required symbol name and press [Return]. Provided the pin names of the component are a subset of the pin names on the chosen symbol, the mapping is valid. You will be warned if there are differences.

Updating the MAP

You can save the mappings you have made by selecting {Update map}. You are prompted for a file name with the extension .MAP which is saved in the net directory. The MAP file is in ASCII and can be edited manually if required. The format is :

"COMPONENT NAME","SYMBOL PATH"

eg "74HC74","C:\BM2\LIB\SYMBOL.LL\DIP14"

This MAP file can be used for automatically mapping netlists on future occasions.

Auto mapping symbols

Once a net list has been loaded and you have previously created a MAP file, you can automatically map the components by selecting {Auto map symbols}. Enter the name for your MAP file to start the process. Any components that are not mapped can be done manually using {Map symbols} and then select {Update map} to add these new mappings.

Making a new PCB

When all symbols are mapped, select {Make PCB} and enter the name you want to give to your board - the



designators, components and nets will then be automatically loaded into the PCB editor. You are then prompted to create a LOG file with file extension .LOG. This text file is saved in the net directory and gives a summary of the creation of the PCB.

Top modifying a PCB

This facility allows you to introduce changes to existing schematic diagrams or netlists at any stage during the design process. By making changes using your schematic capture package, the modifications are processed starting at the top level downwards - the corresponding changes are automatically made to the PCB. Unchanged sections of the PCB are left untouched, requiring only the areas affected by the change to be re-worked. This ensures the changes are only made once in one place and are therefore easier to monitor and importantly, the integrity between schematic and PCB is maintained.

To achieve this, convert and load the netlist as described above. Map the symbols, either manually or using the Auto map feature and select {Top modify PCB}. The PCB file and the new netlist are compared. Any components new to the netlist are added at the bottom of the PCB editor. Any components which were deleted from the netlist/schematic will be deleted from the PCB. All existing tracks, pads, texts and undesignated symbols will not be affected.

In summary :

- ☐ New components are added at the bottom of the screen.
- ☐ Deleted components are removed from the PCB.
- ☐ Tracks, pads and texts are not changed.
- ☐ Undesignated symbols e.g. board outlines, targets etc are not changed.

You then have the option to update the LOG file to record the changes you have made. All new and deleted components are listed as a modification to the PCB along with the date and time.

BOARDMAKER NET LIST FORMAT

```
; This is a comment
; COMPONENT LIST
[R1, AXIAL0.3, 470K]
[R3, AXIAL0.3, 10K]
[U10, DIP8, LM741]
[U11, DIP8, LM741]
; NET LIST
```

```
(
N00001,10,10,10,SIGNAL,NOM-2,MIN-1,VIA-16
R1 {470K}, 2
R3 {10K}, 2
U10 {LM741}, 2 {-INPUT}
)
(
N00002,10,10,10,SIGNAL,NOM-2,MIN-1,VIA-16
R1 {470K}, 1
U11 {LM741}, 3 {+ INPUT}
)
```

%%TRACKS

1-10

2-15

%%ENDTRACKS

%%VIAS

16-35,27

%%ENDVIAS

In general form, this can be described as :

The component list with each component enclosed in square brackets

[Designator(mandatory), Symbol name (mandatory), Value (optional)]

The net list with each enclosed by round brackets. The first line on each net is the net header and is then followed by subsequent net nodes.

(
Net name(mandatory), TT(mandatory), TP(mandatory), PP(mandatory), net type(mandatory), NOM, MIN, VIA Designator, {Value}, Pin name {Alias}
)

The track and via sizes for the net

Track width number - track width in thou

Via width number - diameter in thou, hole size in thou

where :

TT is the track to track design rule clearance in thou

TP is the track to pad design rule clearance in thou

PP is the pad to pad design rule clearance in thou

net type is either UNSPECIFIED, SIGNAL, POWER or BUS

NOM is the nominal track width number (1-8)

MIN is the minimum track width number (1-8)

VIA is the via size number (1-16)

NET ATTRIBUTE EDITOR

Purpose To change the net attributes on a batched basis.

Mode All

Description The net attribute editor provides a means of changing the design rules, track sizes, net type and via size for each of the nets. This can also be achieved by editing each net and changing its attributes, but this method allows you to easily make changes to many if not all of the nets in one pass.

When a PCB file is loaded, select {Net}{Attribute editor} and the nets are loaded into the editor and presented in a scrolling window which is sorted in alphanumeric order. Each net is shown with its design rules, net type, nominal track width, minimum track width and via size.

You then need to mark or tag the nets that you want to edit by moving the banner over the relevant net and pressing [Return]. A check will appear in the left hand column. Selecting it again will untag the net. If you want to tag all the nets press [F2] or [F3] to untag all nets.

Once you have tagged the nets you want to change, press [F1] and a window will present the net attributes. Use the [+] and [-] keys to increment or decrement the attributes to those required for the tagged nets. When you are ready, select the option {Apply these to the tagged nets} and the tagged nets will be changed.

To update the PCB file and exit, press [Esc] and you are prompted whether you wish to exit and save the changes. If so the PCB file will be amended and control is passed back to BoardMaker.

Keys [Ctrl][N]{Attribute editor}

Menu {Net}{Attribute editor}

See also EDIT NET

PAD MATRIX

Purpose To place a set of pads in a matrix

Mode Pad

Description This facility is designed to cater for non-standard component pin pitches. You can specify the repeat count and pitching in both X and Y direction. This is particularly useful for surface mount components.

To use this facility, first add a pad and select {Option}{Matrix} or press [T]. You are then prompted :

Please enter X pitch :

Enter the pitch required in the format [-]999.9999[IN|MM]. Don't type the []'s, they indicate optional items. If you enter the - before the numbers the pads will pitch from right to left (the default is left to right). You can specify the units as either IN or MM, they default to the current UNITS setting. You are then prompted :

Please enter X total :

Enter the total number of pads you require in the X direction. You are then prompted :

Please enter Y pitch :

Enter the pitch required in the format described above. If you want a single line of pads enter 0. You are then prompted :

Please enter Y total :

Enter the total number of pads you require in the Y direction. Enter 1 for a single line of pads.

Keys [Ctrl][O][T]

Menu {Option}{Matrix}



POSTSCRIPT

Purpose

To create an Adobe PostScript file.

Mode

All

Description

In addition to the plotter and printer support provided by BoardMaker, V2.23 includes a driver compatible with the Adobe PostScript language. This may be used to produce artwork on any PostScript compatible device such as an Apple LaserWriter. Equally, when using suitable wordprocessing or desk top publishing package (eg WordPerfect, Ventura Publisher), quality artwork can be incorporated directly into technical documentation.

Plot setup

The {Plot setup} option allows you to determine the type of plot you want to produce.

- ☐ **Pad hole.** You have the choice of whether to fill or avoid the holes in the pads.
- ☐ **Plot mode.** This determines the type of output produced. {Normal} will print all the items, including tracks, pads, text and symbols. {Drilling template} will print the pads only, to be used as a drilling guide. {Solder resist mask} will produce the pads suitably enlarged, with their holes filled in. This can then be photographically reversed and used as the solder resist mask. When you select {Drilling template} the pad holes will be automatically avoided - when you select {Solder resist mask} pad holes will be automatically filled.
- ☐ **Magnification factor.** Use this to change the scale of plot you require. This can be set in the range 0.1 to 10.0 - enter 1.0 for a 1:1 plot. The Drawing details window will display the actual size the plot would be.
- ☐ **Solder resist mask.** This allows you determine the amount of clearance you want the solder resist mask to have for the {Plot mode} setting above.

- ☐ **Encapsulated.** If you intend to import the PostScript file into a publishing program, you will need to use encapsulated PostScript, so set this option to YES. Otherwise, if you are sending it to a PostScript laser printer, make this option NO.
- ☐ **Extra bounding box.** You can choose to have an extra clearance around the artwork when the PostScript file is produced. Select this option if you want to change the bounding box from the default value of one inch.

Choosing the layer to plot

As you select the {Plot layer} option, you increment the layer that will be plotted, both in the menu itself and the Current plot setup window - click on it until the layer you want to plot is displayed.

Output file names

The default output file names are :

- ☐ Layer n : xxxxxx-n.PO1
- ☐ Drilling template : xxxxxx-D.PO1
- ☐ Top solder resist mask : xxxxxx-T.PO1. Make sure you select Solder resist mask and layer 1 to generate this output.
- ☐ Bottom solder resist mask : xxxxxx-B.PO1. Make sure you select Solder resist mask and layer 8 to generate this output.

where xxxxxx are the first six characters of the PCB file name and n is the layer number. eg SAMPLE-1.PO1 or ROUTED-D.PO1

Save setup

This option lets you save the settings you have made.

See also

PRINT, PHOTOPLOT, PENPLOT BLOCK, NC DRILL

REPEAT Block

Purpose To repeat the current block and its contents.

Mode Block

Description This command provides a method of copying the contents of a block to a new position on the drawing. The affected items in the block can be selected by the user and can be any combination of pads, tracks, text and symbols. Consequently, using the REPEAT command in block mode is the most comprehensive method of copying existing items.

To select which items in the block are affected by this command, use the BLOCK OPTIONS menu in the {Config} screen. This allows you to select whether the REPEAT command will pickup or leave tracks, pads, text or symbols that are enclosed in the block.

Note that if the CAPTURE MODE is set to rubberbanding on the block cannot be repeated and you are warned to change the capture mode. and a block cannot be copied directly onto itself.

By selecting {Config}{Block settings}{Copy mode}, symbols in the block which have designators can either be directly REPEATED or INCREMENTED. The INCREMENT option allows you to create new unique designators, the REPEAT option is useful when you have completed a board and want to step and repeat it ready for manufacture.

Keys [R]

Menu {Option}{Repeat}

See also SHIFT, MIRROR, BLOCK ORIGIN, CAPTURE MODE, KILL, ANTICLOCKWISE

RENAME SYMBOL

Purpose To rename a symbol in a PCB or Schematic editor

Mode All

Description Using this command, you can rename a symbol from within the PCB or Schematic editor. This may be convenient in some cases when you want to use the UPDATE SYMBOL facility but the symbol names are not the same.

Select {Library}{Rename symbol} and the {Option} menu will display a list of all the symbols in the current drawing. Select the one you want to change and you will be prompted:

Editing symbol name: DIP14

Simply enter the new name for the symbol. Existing symbol names are not valid and cannot be entered.

Keys [Ctrl][L]{Rename symbol}

Menu {Library}{Rename symbol}

See also UPDATE SYMBOLS, EXPORT SYMBOL

REPORTS

Purpose To create a PCB report output

Mode All

Description The purpose of the reports output is to provide all the information about a PCB which you can then use for documentation. When you select {File}{Output..}{Reports} you are presented with the Report generator screen and have the option to output the following reports :

Statistics

This gives the total number of tracks, pads, texts, symbols, segments, characters, nets and net nodes on the PCB. Also the size of the board is reported.

Full PCB description

Describes the position and layer of all tracks, pads, symbols and texts on the PCB.

Component details

Gives a list of all components on the PCB, its symbol name, value (if any) and position relative to the fixed origin.

Bill of Materials

Collates the components on the board by value and symbol name and gives the number of each along with the component designators. Components with values are sorted first, those without are sorted last.

Database ASCII

This produces the same information as the component details output, but is suitable for import into spreadsheet and database software. The data are delimited by quotes and separated by commas as follows :

"C2","AXIAL0.3","470uF 25V",12.56,3.12

The X, Y position is numeric and as such is not delimited by quotes.

Output to...

You can choose to output to file, LPT1, LPT2, COM1 or COM2.

Page length

This allows you to set the page length in lines to suit your printer/paper requirements. The default value is 66 lines.

To produce the report, click on the options you want reporting until they show YES, set the page length, output port/file and then select {Generate report} when ready.

Keys

[Ctrl][F]{Output..}{Reports}

Menu

{File}{Outputs..}{Reports}

RESTORE BACKUP

Purpose

To load a previous version of a file

Mode

All

Description

When a PCB or Schematic file is saved, BoardMaker allows you to have up to 9 backup copies (see BACKUP COPIES). Restore Backup provides a means of loading one of these previous versions.

Select {File}{Restore Backup} and you are prompted for a filename. Enter the name required or press [Return] and chose the name from those displayed in the {Option} menu. You are then prompted for the backup file number 1-9.

Restore from which backup : 123.....?

The prompt will show the numbers of the backups which exist. Enter the number required and the file will load. This can then be saved under a new name if required.

Keys

[Alt][R]

Menu

{File}{Restore backup}

See also

BACKUP COPIES, SAVE